
Digital Circuit Design Using Xilinx ISE Tools

Table of Contents

1. Introduction
2. Programmable logic devices: FPGA
3. Creating a new project in Xilinx ISE
 - 3.1 Opening a project
 - 3.2 Creating an Verilog input file for a combinational logic design
 - 3.3 Editing the Verilog source file
4. Compilation and Implementation of the Design
5. Functional Simulation of Combinational Designs
 - 5.1 Adding the test vectors
 - 5.2 Simulating and viewing the simulation result waveforms
 - 5.3 Saving the simulation results
6. Preparing and downloading bitstream for the Spartan FPGA
7. Testing a Digital logic circuit
 - 7.1 Observing the outputs using the on-board LEDs and Seven Segment Display
8. Design and Simulation of sequential circuits using Verilog
 - 9.1 Design of Sequential Circuits
 - 9.2 Simulation of Sequential Circuits
9. Design and Simulation of Finite State machines in Verilog
10. Hierarchical circuit design using Modules
11. Post synthesis Timing simulation with Modelsim.

Appendix:

- A. Verilog Hardware Modeling – Introduction to the Verilog Language.
- B. Digilent BASYS board.

1. Introduction

Xilinx Tools is a suite of software tools used for the design of digital circuits implemented using Xilinx *Field Programmable Gate Array (FPGA) or Complex Programmable Logic Device (CPLD)*. The design procedure consists of (a) design entry, (b) compilation and implementation of the design, (c) functional simulation and (d) testing and verification. Digital designs can be entered in various ways using the above CAD tools: using a schematic entry tool, using a hardware description language (HDL) – Verilog or VHDL or a combination of both. In this lab we will only use the design flow that involves the use of Verilog HDL.

The CAD tools enable you to design combinational and sequential circuits starting with Verilog HDL design specifications. The steps of this design procedure are listed below:

1. Create Verilog design input file(s) using template driven editor.
2. Compile and implement the Verilog design file(s).
3. Create the test-vectors and simulate the design (functional simulation) without using a PLD (FPGA or CPLD).
4. Assign input/output pins to implement the design on a target device.
5. Download bitstream to an FPGA or CPLD device.
6. Test design on FPGA/CPLD device

A Verilog input file in the Xilinx software environment consists of the following segments:

- **Header:** module name, list of input and output ports.
- **Declarations:** input and output ports, registers and wires.
- **Logic Descriptions:** equations, state machines and logic functions.
- **End:** endmodule

All your designs for this lab must be specified in the above Verilog input format. Note that the *state diagram* segment does not exist for combinational logic designs.

Software used: We will use **Xilinx Tools** (v13.3) and **Digilent Adept**.(v2.2.0) in this lab.

Hardware : We will use a BASYS2 kit from Digilent Inc. (with a Spartan FPGA from Xilinx) to implement the designs.

2. Programmable Logic Device: *FPGA*

In this lab digital designs will be implemented in the Pegasus board which has a Xilinx Spartan 3E FPGA XC3S250E. This FPGA part belongs to the Spartan family of FPGAs. These devices come in a variety of packages. We will be using devices that are packaged in 208 pin package with the following part number: XC2S50-PQ208. This FPGA is a device with about 250K gates. Detailed information on this device is available at the Xilinx website.

3. Creating a New Project

Xilinx Tools can be started by clicking on the Project Navigator Icon on the Windows desktop. This should open up the Project Navigator window on your screen. This window shows (see [Figure 1](#)) the last accessed project.

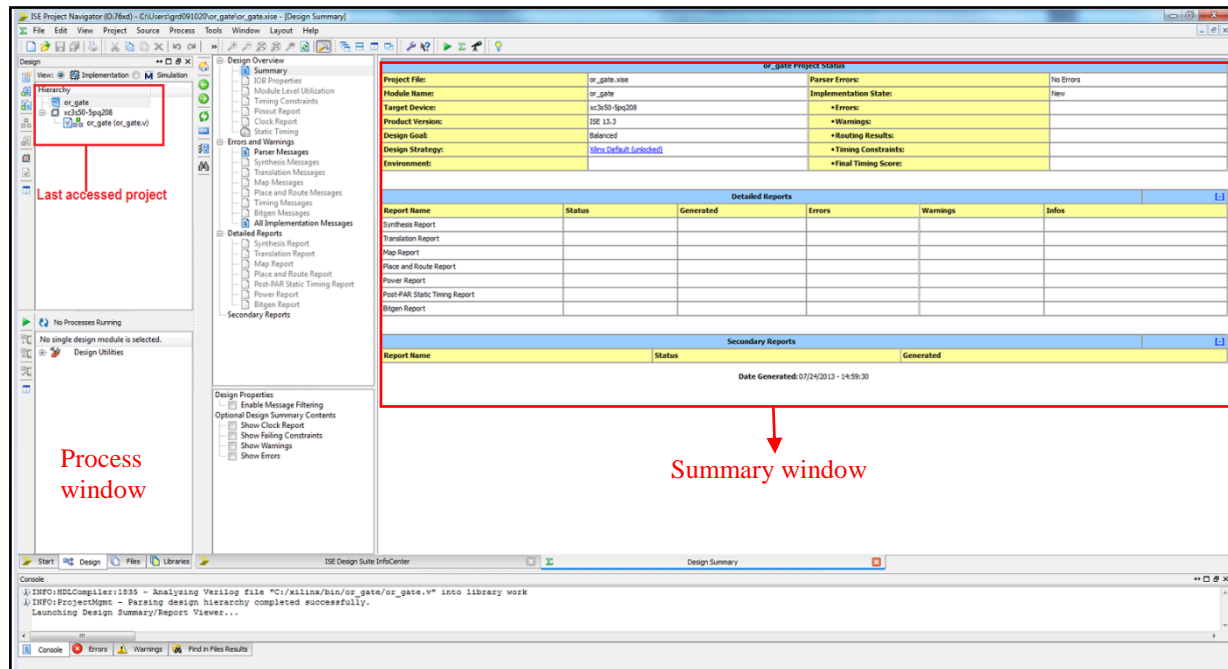


Figure 1: Xilinx ISE Project Navigator window (snapshot from Xilinx ISE software)

3.1 Opening a project

Select **File->New Project** to create a new project. This will launch the **New Project Wizard** ([Figure 2](#)) on the desktop. Fill up the necessary entries as follows:

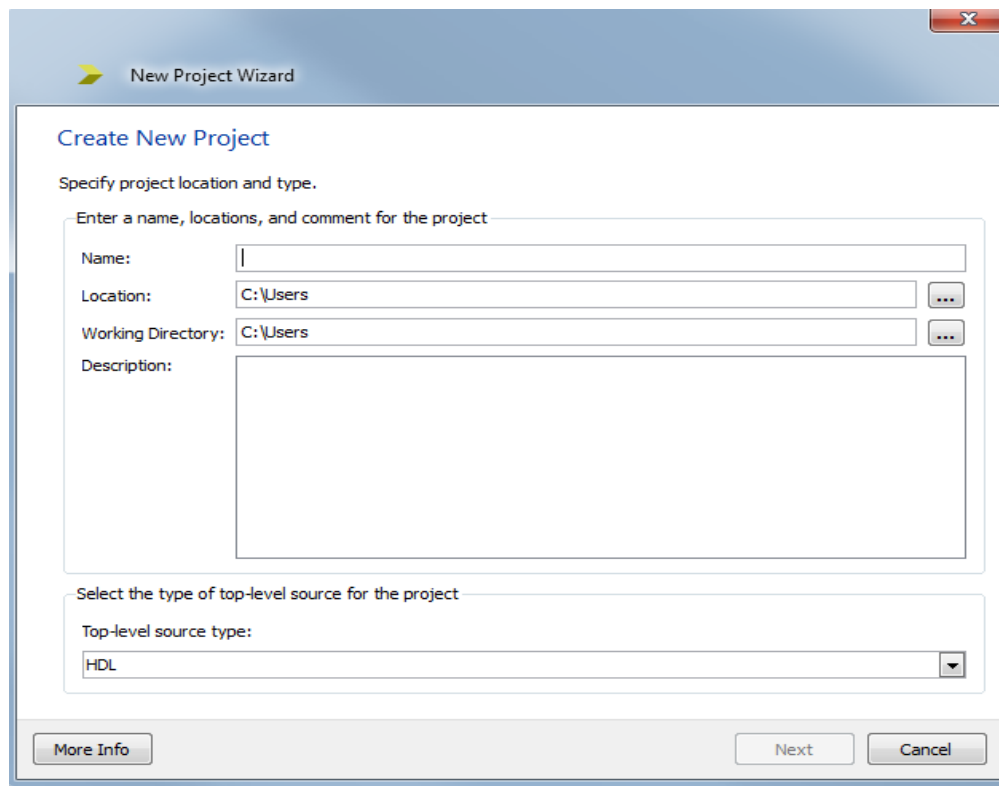


Figure 2: New Project Wizard (snapshot from Xilinx ISE software)

- **Name:** Write the name of your new project (E.g.: or_gate)
- **Location:** The directory where you want to store the new project
- **Working Directory:** The directory where all your project related files will be saved.
- **Description:** (Optional) A brief description of your project

Leave the top level module type as “HDL”.

Example: If the project name were “or_gate”, enter “or_gate” as the project name and then click “Next”.

Clicking on NEXT should bring up the **Project Settings** window:

Project Settings

Specify device and project properties.
Select the device and design flow for the project

Property Name	Value
Product Category	All
Family	Spartan3E
Device	XC3S250E
Package	CP132
Speed	-4
Top-Level Source Type	HDL
Synthesis Tool	XST (VHDL/Verilog)
Simulator	Modelsim-SE Verilog
Preferred Language	Verilog
Property Specification in Project File	Store all values
Manual Compile Order	<input type="checkbox"/>
VHDL Source Analysis Standard	VHDL-93
Enable Message Filtering	<input type="checkbox"/>

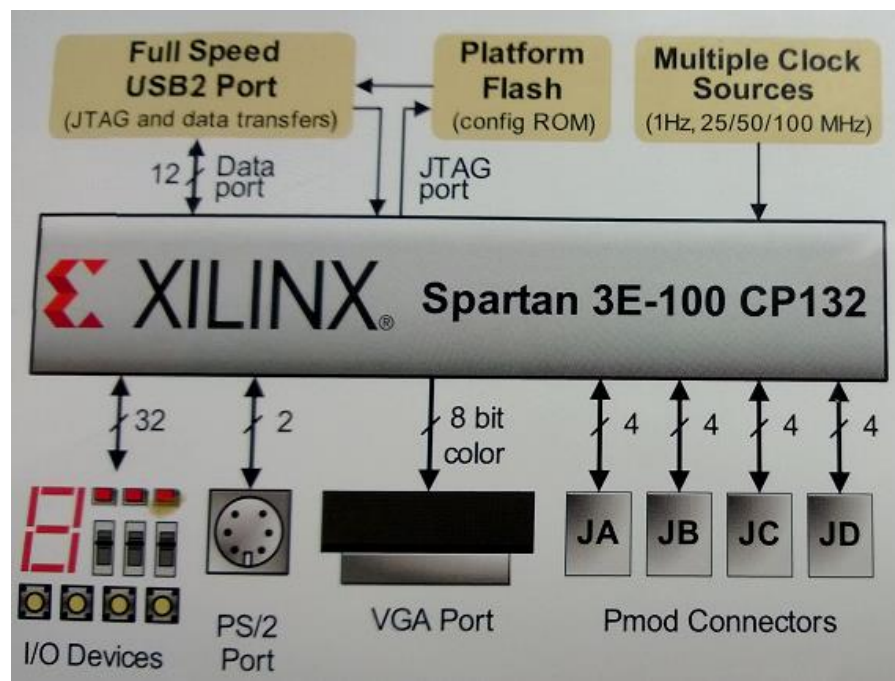
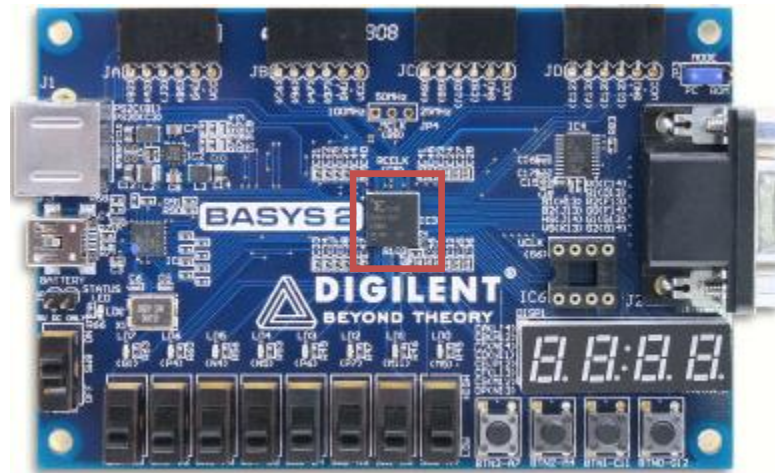
More Info Next Cancel

Figure 3: New Project Wizard- Project settings

For each of the properties given below, click on the ‘**value**’ area and select from the list of values that appear.

- **Device Family:** Family of the FPGA used. In this laboratory we will be using the Spartan3E FPGAs

- **Device:** The number of the FPGA device. Note the device number on the FPGA on the Digilent board! If you are unsure –ask the TA!



Check your board for the device number

- **Package:** The type of package with the number of pins. The Spartan FPGA used in this lab is packaged in C6DGO (which is equivalent to CPG132) package. If you are unsure, ask the T.A!
- **Speed :** The Speed grade is “-4”.

- **Synthesis Tool:** **XST** [VHDL/Verilog]
- **Simulator:** The tool used to simulate and verify the functionality of the design. Choose **ISim** as the simulator.

Then click on **NEXT** to save the entries. The Project Summary window ([Figure 4](#)) will show you a summary of your project details. In this example we have used the project name “**or_gate**”. **Pay attention to the highlighted details in particular!**

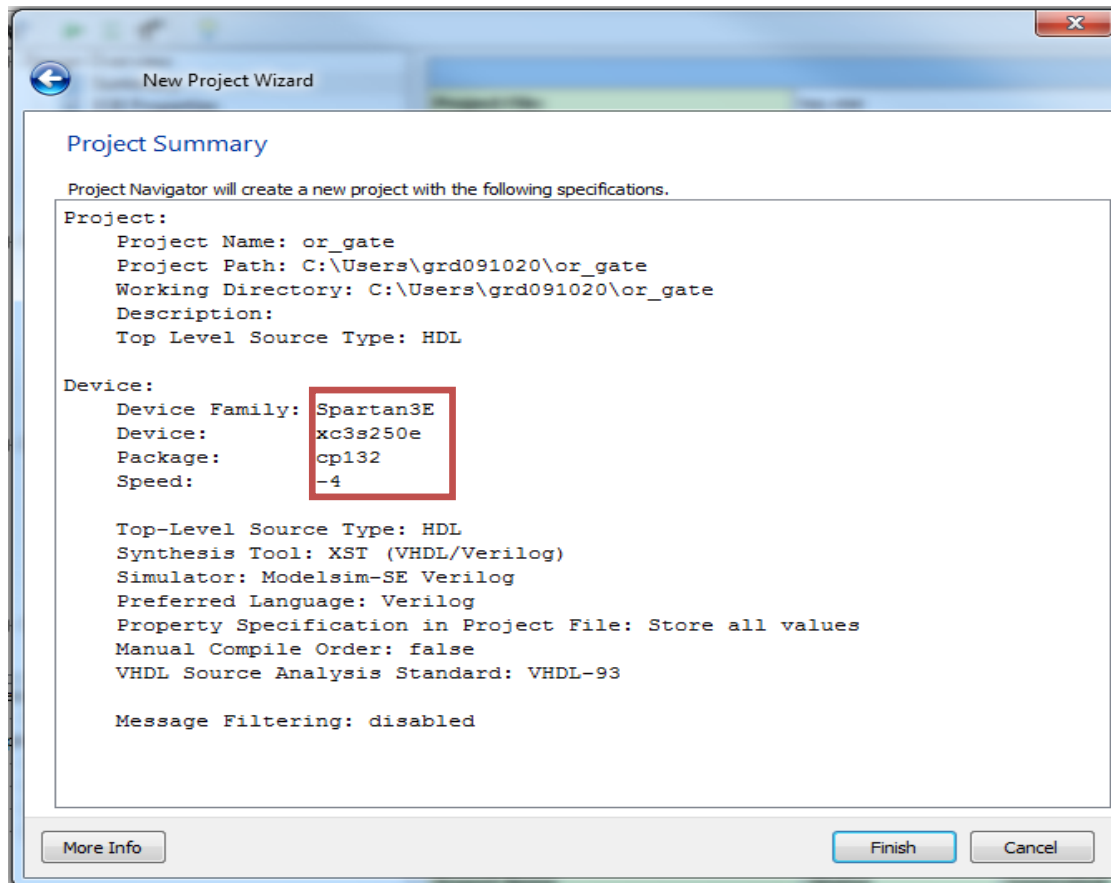


Figure 4: New Project Wizard- Project Summary (snapshot from Xilinx ISE software)

Click on ‘**Finish**’.

All project files such as schematics, netlists, Verilog files, VHDL files, etc., will be stored in a subdirectory with the project name. A project can only have one top level HDL source file (or schematic). Modules can be added to the project to create a modular, hierarchical design (see [Section 9](#)).

In order to open an existing project in Xilinx Tools, select **File->Open Project** to show the list of projects on the machine. Choose the project you want and click **OK**.

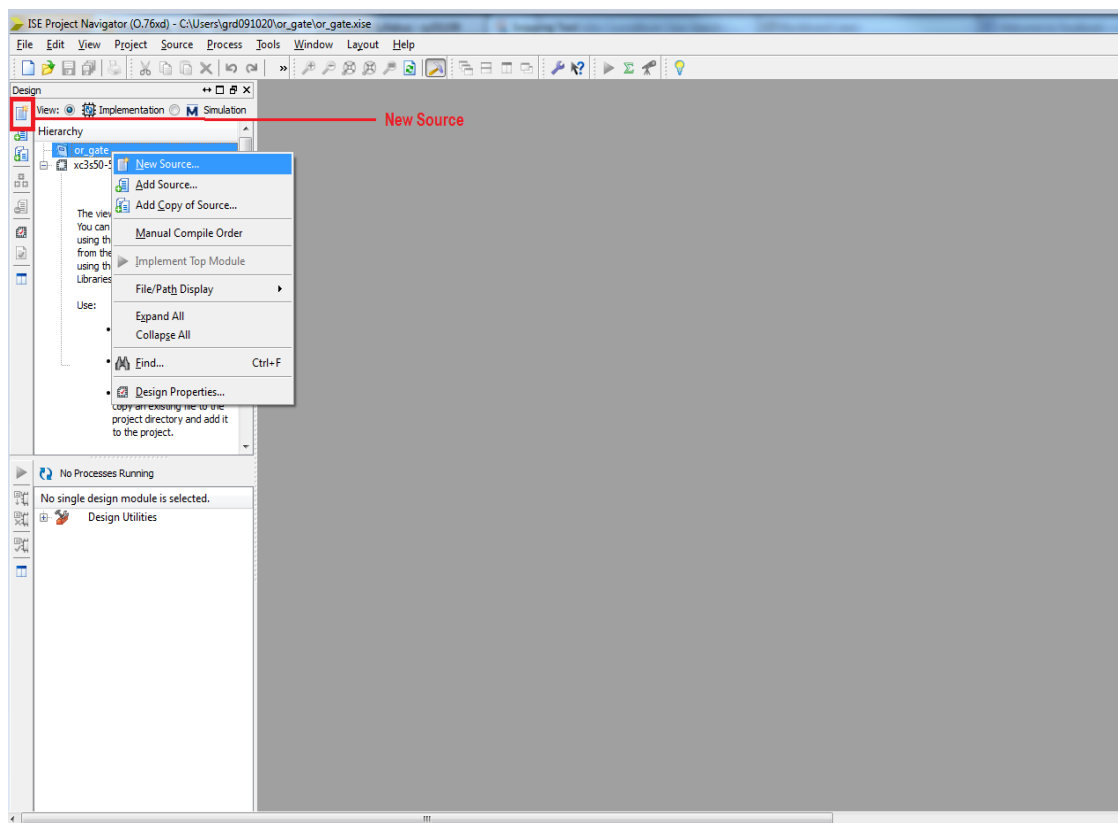


Figure 5: Create New source window (snapshot from Xilinx ISE software)

If you would like to create a new source file, from the ISE window, right click on the project name and choose ‘**New source**’ or click on the **New source** button on the toolbar as shown in **Figure 5**. You may also choose **Project-->New Source** from the menu.

3.2 Creating a Verilog HDL input file for a combinational logic design

In this lab we will enter a design using a structural or RTL description using the Verilog HDL. You can create a Verilog HDL input file (.v file) using the HDL Editor available in the Xilinx ISE Tools (or any text editor).

If adding an already existing source file (.v file) to the project, in the project Navigator window, select **Project -> Add Copy Source** and browse through the disk for the source file.

If creating a new source file, in the Project Navigator window, select **Project -> New Source**. A window pops up as shown in [Figure 6](#). (Note: “**Add to project**” option is selected by default. If you do not select it then you will have to add the new source file to the project manually.)

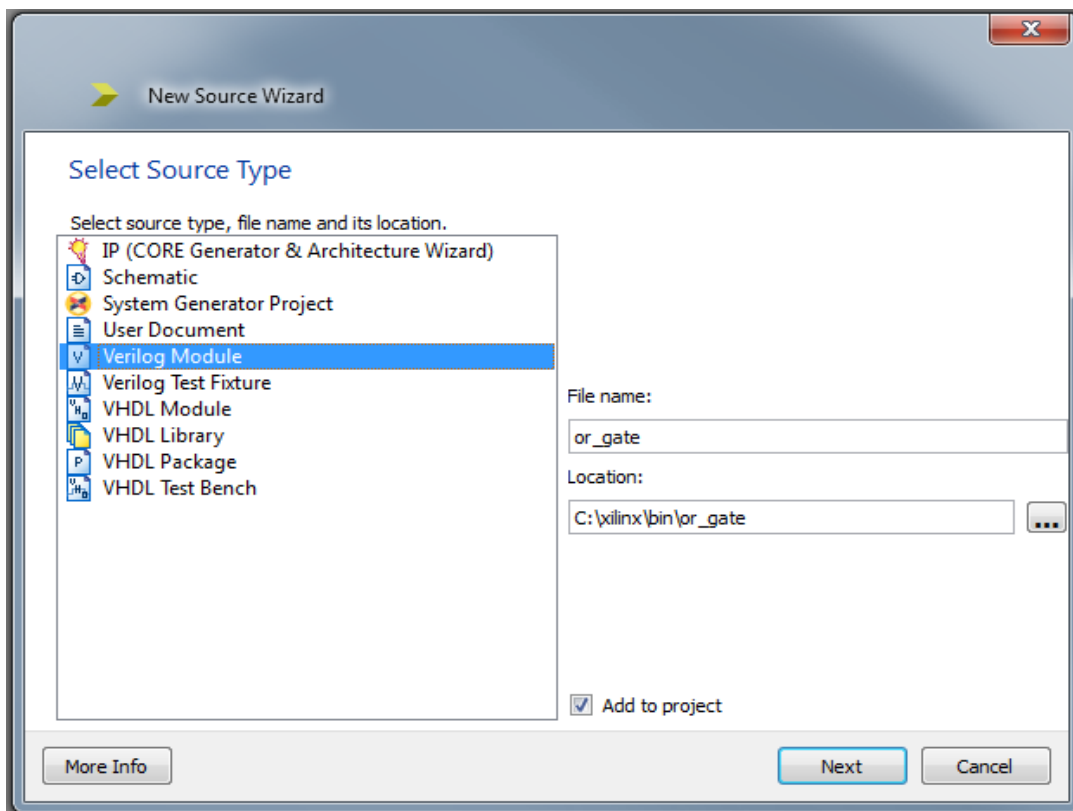


Figure 6: New Source Wizard -creating Verilog-HDL source file (snapshot from Xilinx ISE software)

Select **Verilog Module** and in the “File Name:” area, enter the name of the Verilog source file you are going to create. Also make sure that the option **Add to project** is selected so that the source need not be added to the project again. Then click on **Next** to accept the entries. This pops up the following window ([Figure 7](#)).

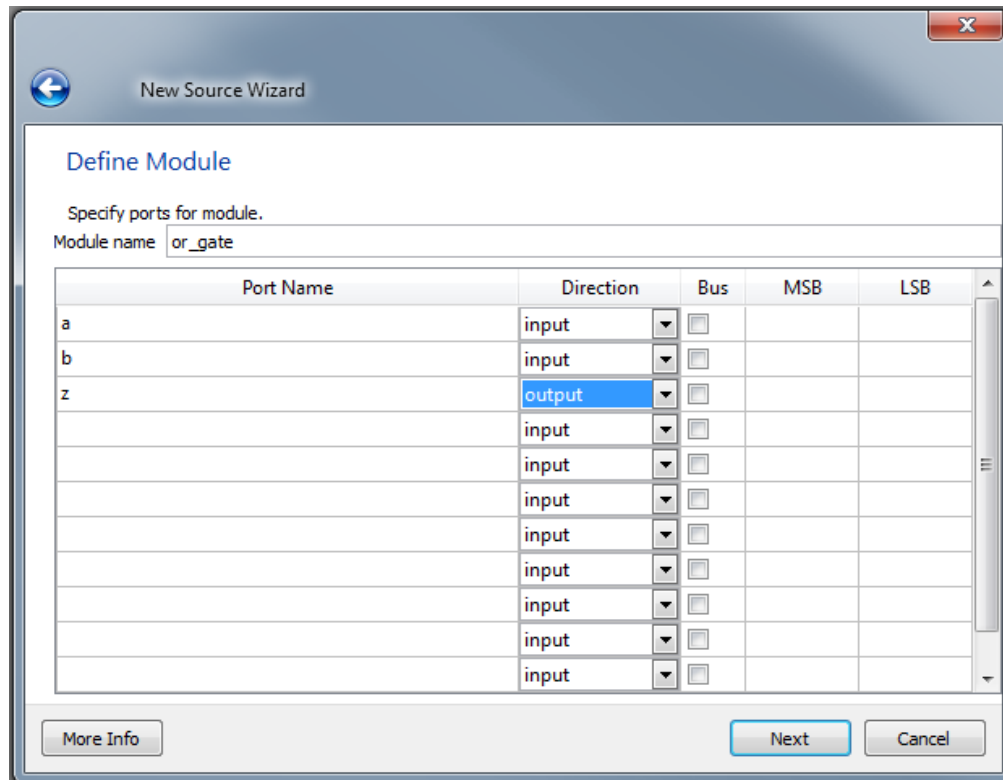


Figure 7: Define Verilog module window (snapshot from Xilinx ISE software)

In the **Port Name** column, enter the names of all input and output pins and specify the **Direction** accordingly. A Vector/Bus can be defined by entering appropriate bit numbers in the **MSB/LSB** columns. Then click on **Next**> to get a window showing all the new source information ([Figure 8](#)).

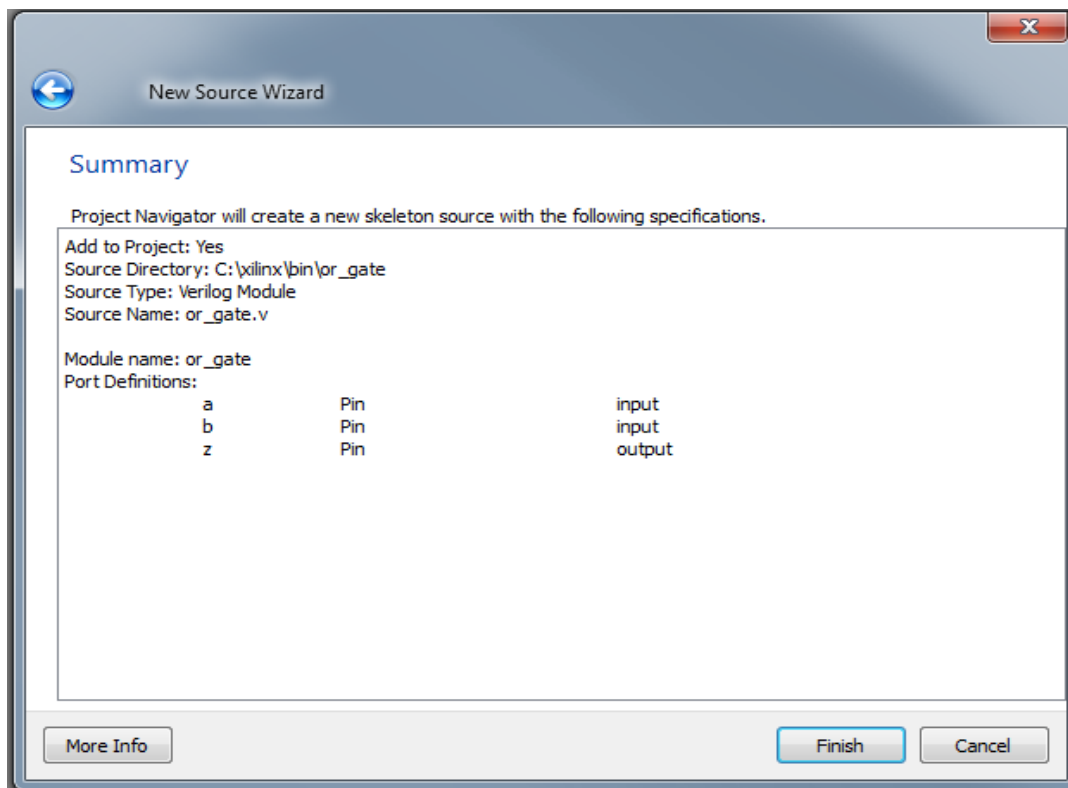


Figure 8: New Source Wizard window(snapshot from Xilinx ISE software)

Once you click on **Finish**, the source file will be displayed in the sources window in the **Project Navigator** ([Figure 1](#)).

If a source has to be removed, just right click on the source file in the **Sources in Project** window in the **Project Navigator** and select **Remove** in that. Then select **Project -> Delete Implementation Data** from the Project Navigator menu bar to remove any related files.

3.3 Editing the Verilog source file

The source file will now be displayed in the **Project Navigator** window ([Figure 9](#)). The source file window can be used as a text editor to make any necessary changes to the source file. All the input/output pins will be displayed. Save your Verilog program periodically by selecting the **File->Save** from the menu. You can also edit Verilog programs in any text editor and add them to the project directory using “Add Copy Source”.

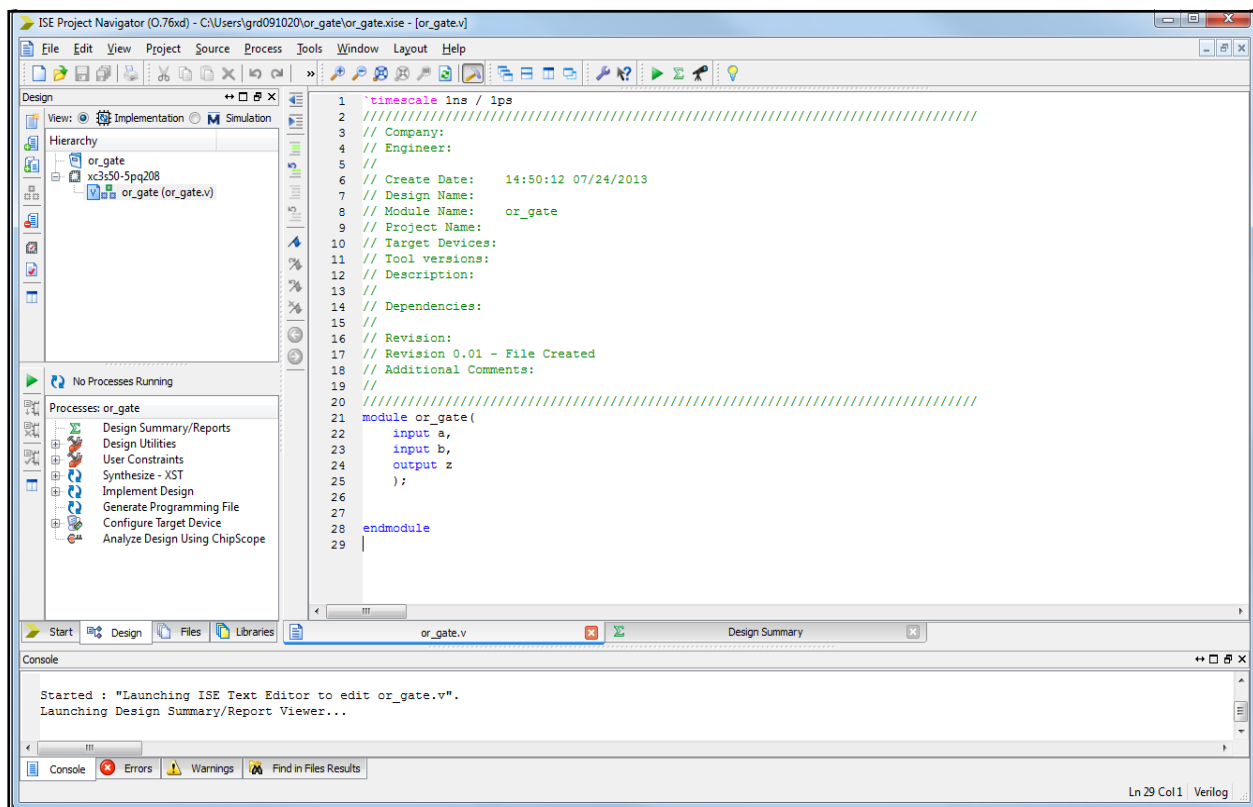


Figure 9: Verilog Source code editor window in the Project Navigator (from Xilinx ISE software)

- **Adding Logic in the generated Verilog Source code template:**

A brief Verilog Tutorial is available in Appendix-A. For the language syntax and for construction of logic equations please refer to Appendix-A.

The Verilog source code template generated shows the module name, the list of ports and also the declarations (input/output) for each port. Combinational logic code can be added to the verilog code after the declarations and before the **endmodule** line.

For example, an output **z** in an OR gate with inputs **a** and **b** can be described as,

$$\text{assign } z = a / b;$$

Remember that the variable names are case sensitive.

- **Other constructs for modeling a logic function:**

A given logic function can be modeled in many ways in verilog. Verilog offers numerous constructs to efficiently model designs. Here is an example in which a 4x1 multiplexer, is implemented using a case statement:

```

module mux4x1 (a,b,c,d,sel,y);
  input a, b, c, d;
  input [1:0] sel;
  output y;
  reg y;
  always @ (a or b or c or d or sel)
  case (sel)
    0 : y = a;
    1 : y = b;
    2 : y = c;
    3 : y = d;
    default : y=1'b0; //One bit binary value '0'
  endcase
endmodule

```

In the example shown, “sel” is the 2 bits wide select input to the multiplexer. The ‘default’ case in the case statement is reached when the variable value does not match any of the case options.

The 4x1 multiplexer can also be implemented using a set of if-then-else constructs as follows:

```





module mux4x1 (a,b,c,d,sel,y);
  input a, b, c, d;
  input [1:0] sel;
  output y;
  reg y;
  always @ (a or b or c or d or sel)
  if(sel == 2'b00)
    y=a;
  else if (sel ==2'b01)
    y=b;
  else if (sel ==2'b10)
    y=c;
  else if (sel ==2'b11)
    y=d;
  else
    y=1'b0;
endmodule

```

Even though both the code segments shown above realize a 4x1 multiplexer, each one of them leads to a different hardware implementation for the given logic. Explaining the hardware implementations for specific Verilog construct is beyond the scope of this document. Interested readers can refer to the links in the Verilog tutorial available in Appendix-A.

4. Compilation and Implementation of the Design

The design has to be compiled and implemented before it can be checked for correctness, by running functional simulation or downloaded onto the prototyping board. With the top-level Verilog file opened (can be done by double-clicking that file) in the HDL editor window in the right half of the Project Navigator, and the view of the project being in the **Module view**, the **Implement Design** option can be seen in the **Process Window**. **Design Entry Utilities** and **Generate Programming File** options can also be seen in the process view. The former can be used to include user constraints, if any and the latter will be discussed later.

To compile the design, expand the Implement Top Module (Figure 10) by clicking on **Process->Implement Top Module** OR by right clicking on the design and choosing Implement Top Module. It will go through steps like **Check Syntax**, **Compile Logic**, **Interpret Feedbacks**, **Reformat Logic** and **Optimize Hierarchy**. If any of these steps could not be done or done with errors, it will place a  mark in front of that, otherwise a tick mark  will be placed after each of them to indicate the successful completion. If everything is done successfully, a  mark will be placed before the **Synthesize-XST** option. If there are warnings, one can see  mark in front the **Console** window present at the bottom of the Navigator window. *Every time the design file is saved; all these marks disappear asking for a fresh compilation.*

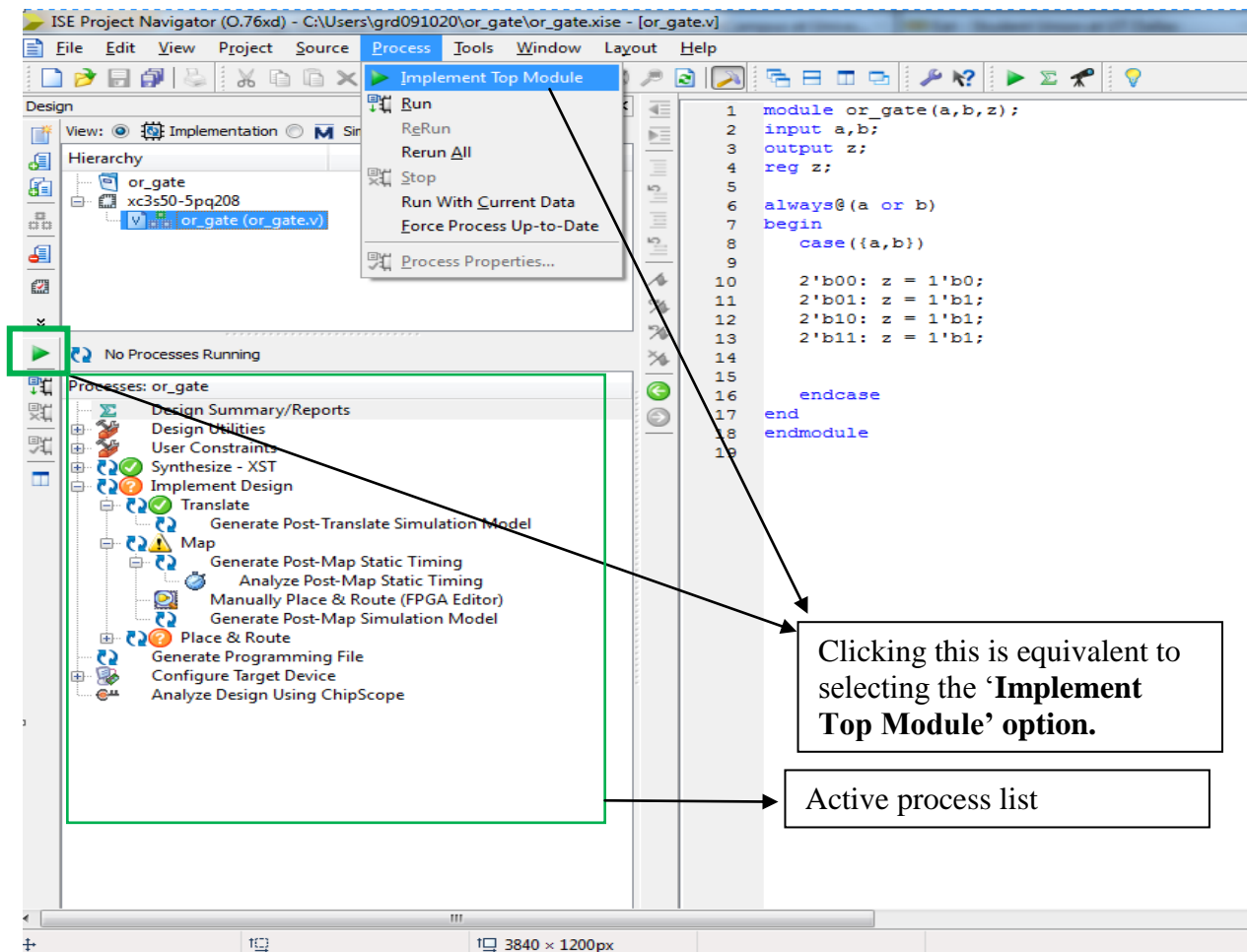


Figure 10: Implement Top Module option in Xilinx ISE

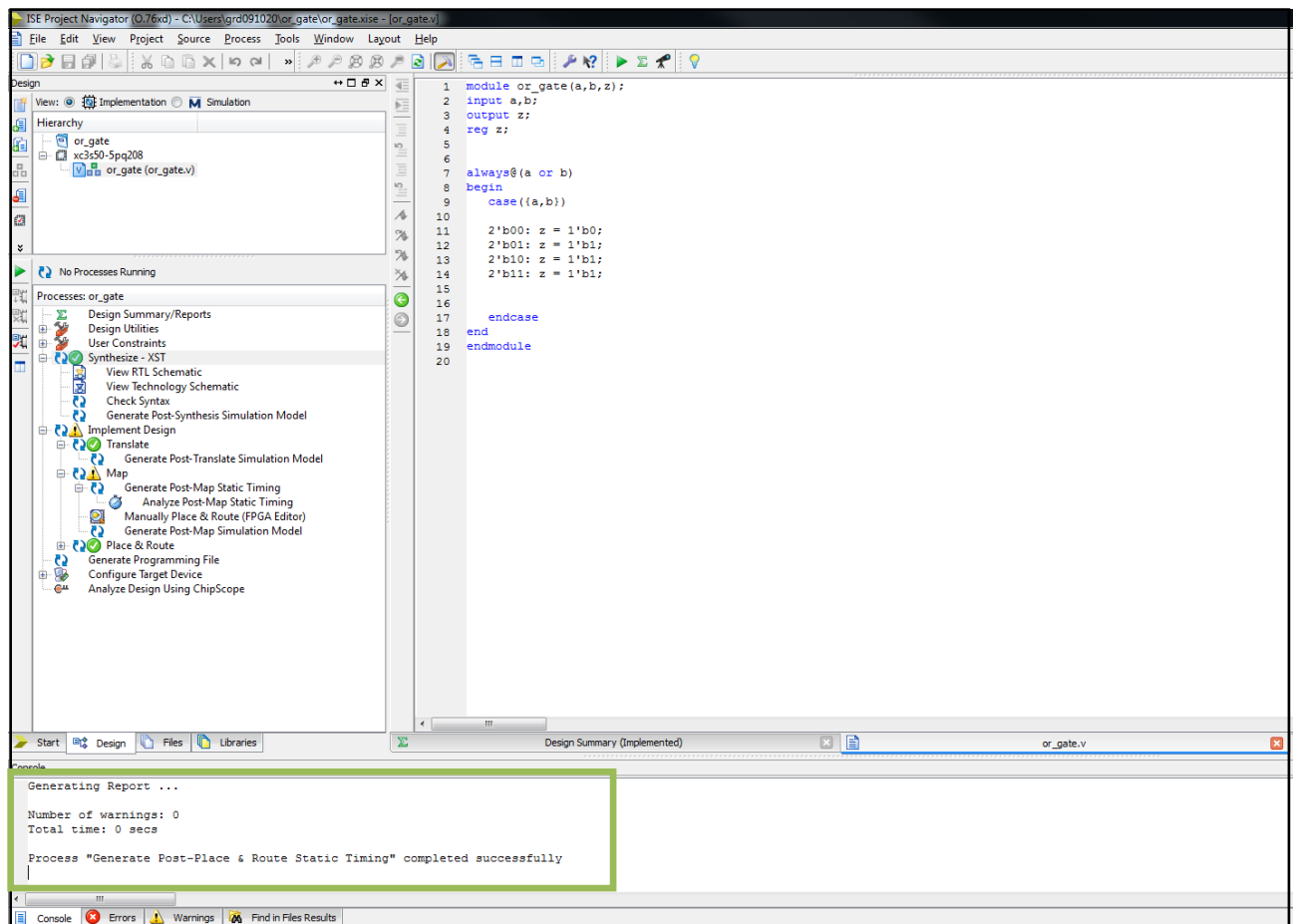


Figure 12 : Implementing the Design (snapshot from Xilinx ISE software)

If there are no errors then the tool shows that the design has been synthesized successfully as in [Figure 12](#).

5. Functional Simulation of Combinational Designs

5.1 Creating a Testbench

To check the functionality of a design, we have to apply test vectors and simulate the circuit. In order to apply test vectors, a **testbench** file is written. Essentially it will supply all the inputs to the module designed and can check the outputs of the module. Example: For the 2 input OR Gate, the test bench is created as follows:

1. Right click on the design (or_gate) and select **New Source**

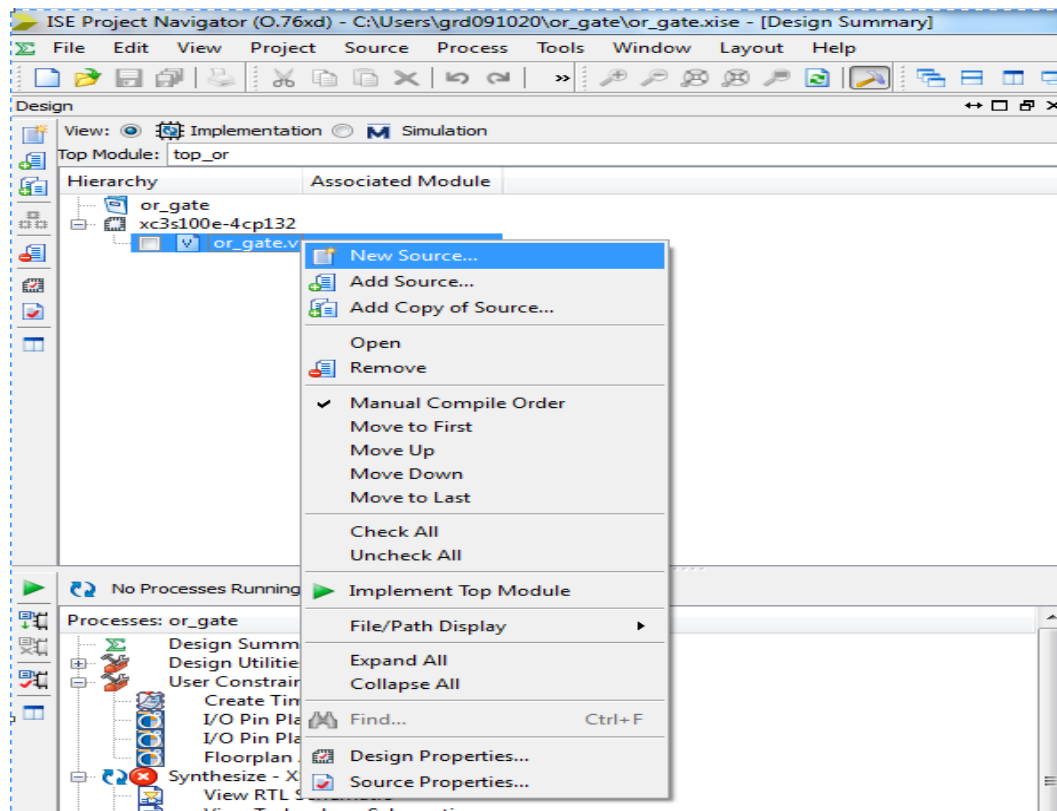


Figure 13 : Adding a test bench to your design-step1

2. From the new source wizard pop up, select **Verilog Test Fixture** and enter a name for the test bench (eg: or_tb) in the File name field as shown in the Figure and click Next.

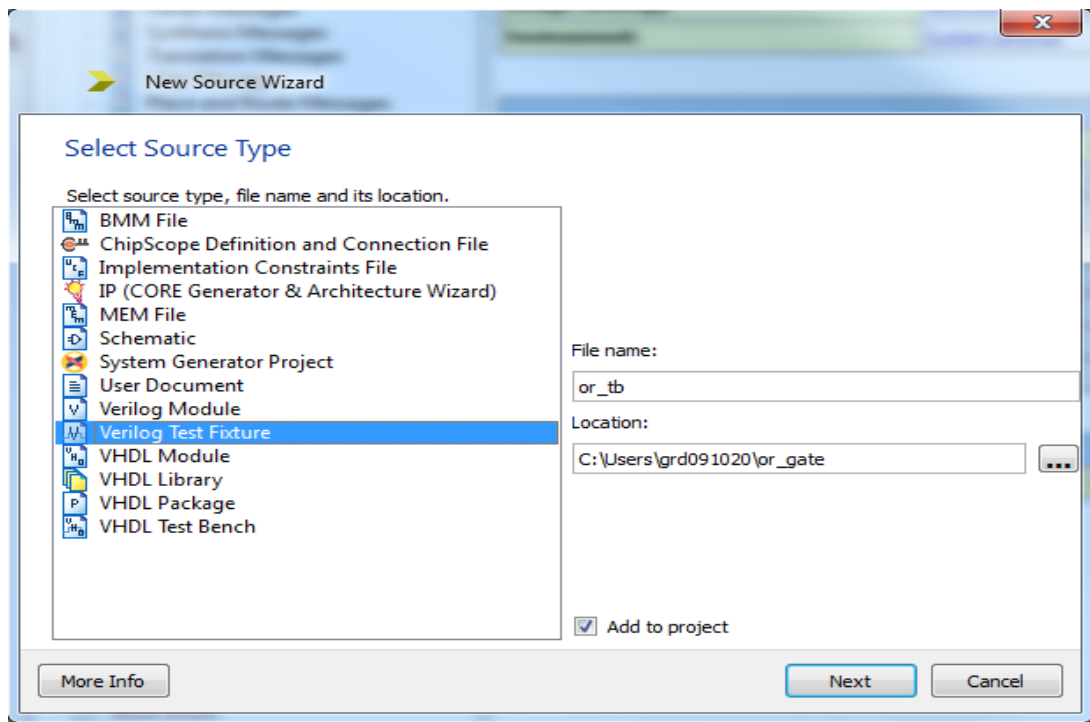


Figure 14 : Adding a test bench to your design-step 2

3. The New source wizard will show the design that the test bench is associated with (this should be the design that you wish to simulate- in this case, or_gate). Click **Next** and then **Finish**.

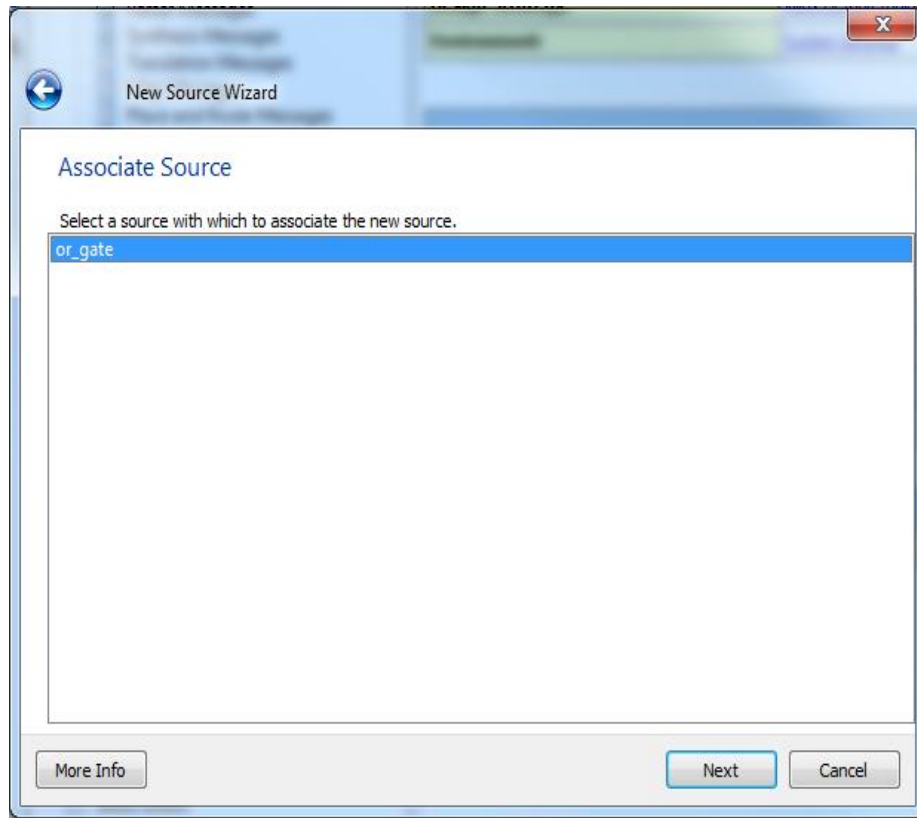


Figure 15 : Adding a test bench to your design-step 3

4. The ISE project navigator will generate some boilerplate test bench code for you – you may either modify this code or replace it with the following code. Refer to the Figure

```

module or_tb(a,b,z);
output a;
output b;
input z;

reg a,b; //declaration that a and b are registers

// Instantiate the Unit Under Test (UUT)

or_gate uut (

    .a(a),
    .b(b),
    .z(z)
);

initial
begin

```

```

//test stimuli
a <= 1'b0;
b <= 1'b0;
#100;
a <= 1'b0;
b <= 1'b1;
#100;
a <= 1'b1;
b <= 1'b0;
#100;
a <= 1'b1;
b <= 1'b1;
#100;
end
endmodule

```

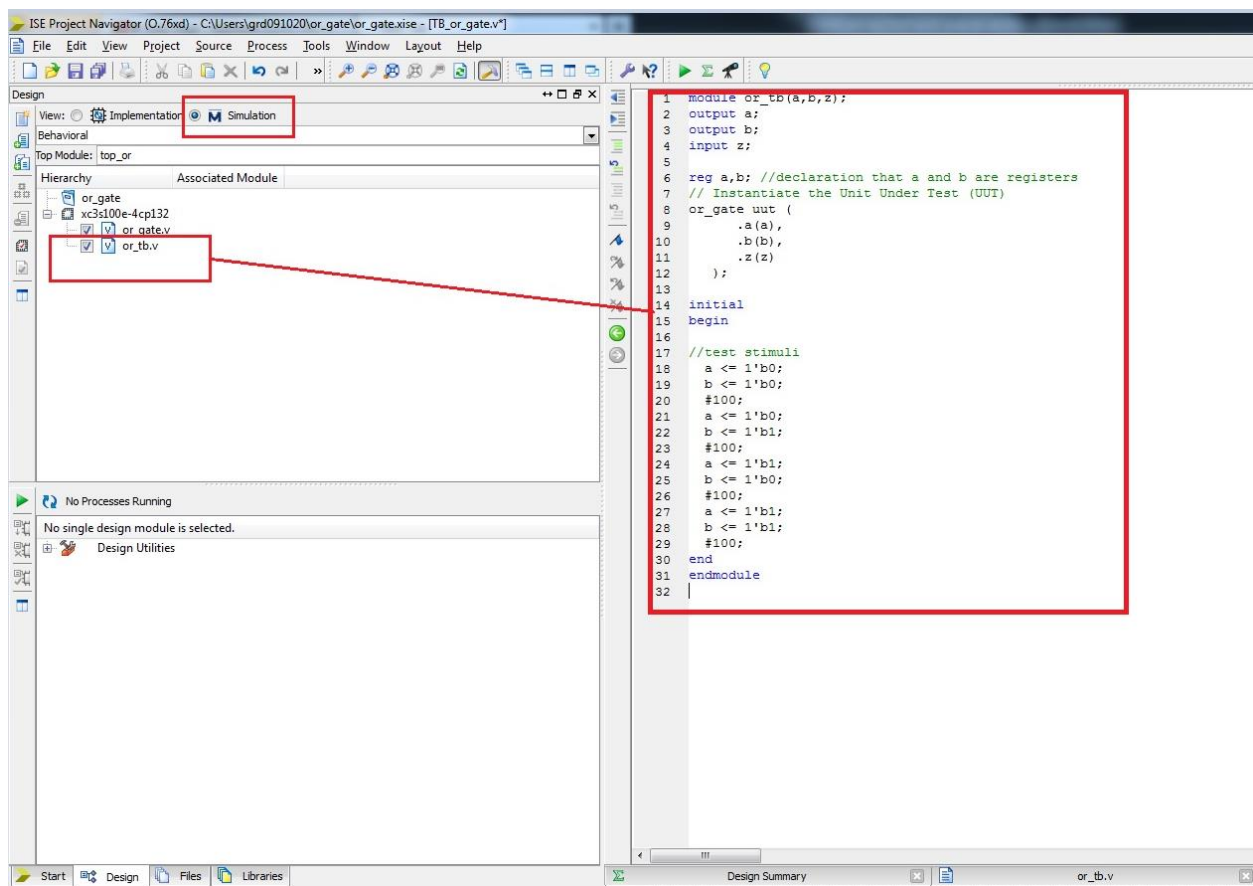


Figure 16: Simulation window in ISE showing the newly created test bench.

5.2 Simulating and Viewing the Output Waveforms

In the process window right click on Simulate Behavioral Model and click on Run.

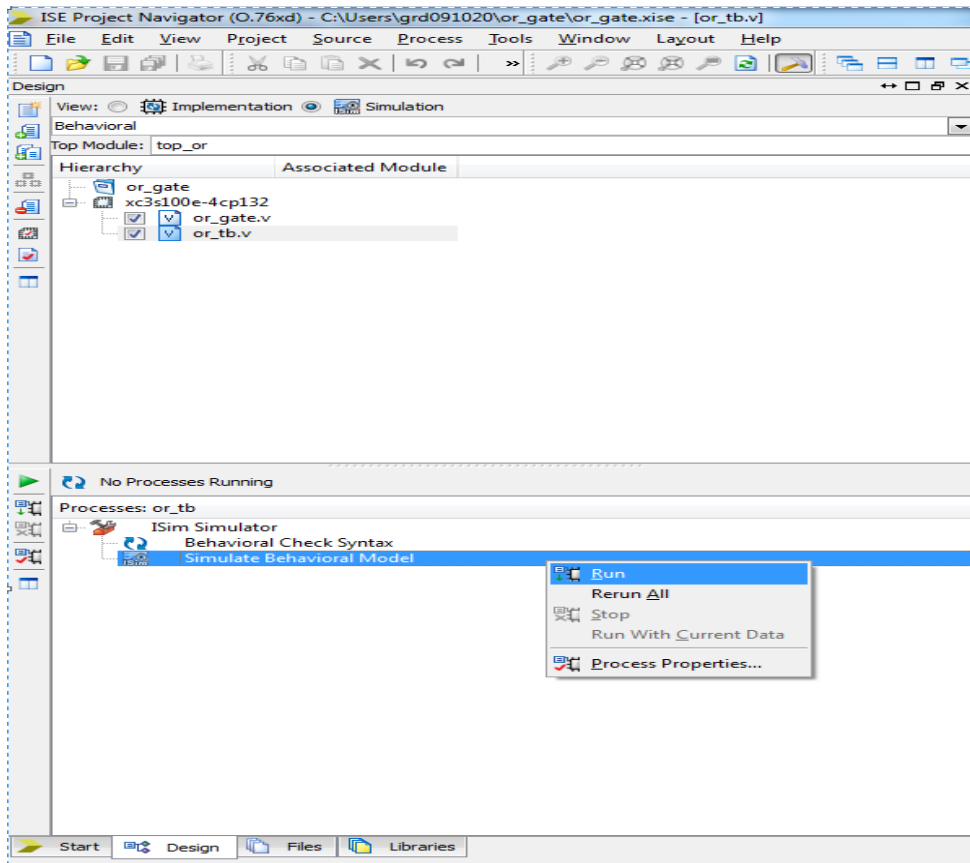


Figure 17 : Launching the iSIM simulator from Project Navigator

If the code is free of syntax errors, this will launch the ISim simulator window as shown in figure (Note; In case you encounter an error when trying to launch the simulator, right click on the design and ensure that “manual compile order” is not checked). Also make sure that the test bench passes the syntax checks. You will be able to observe the simulation waveforms as per the test stimulus in this window.

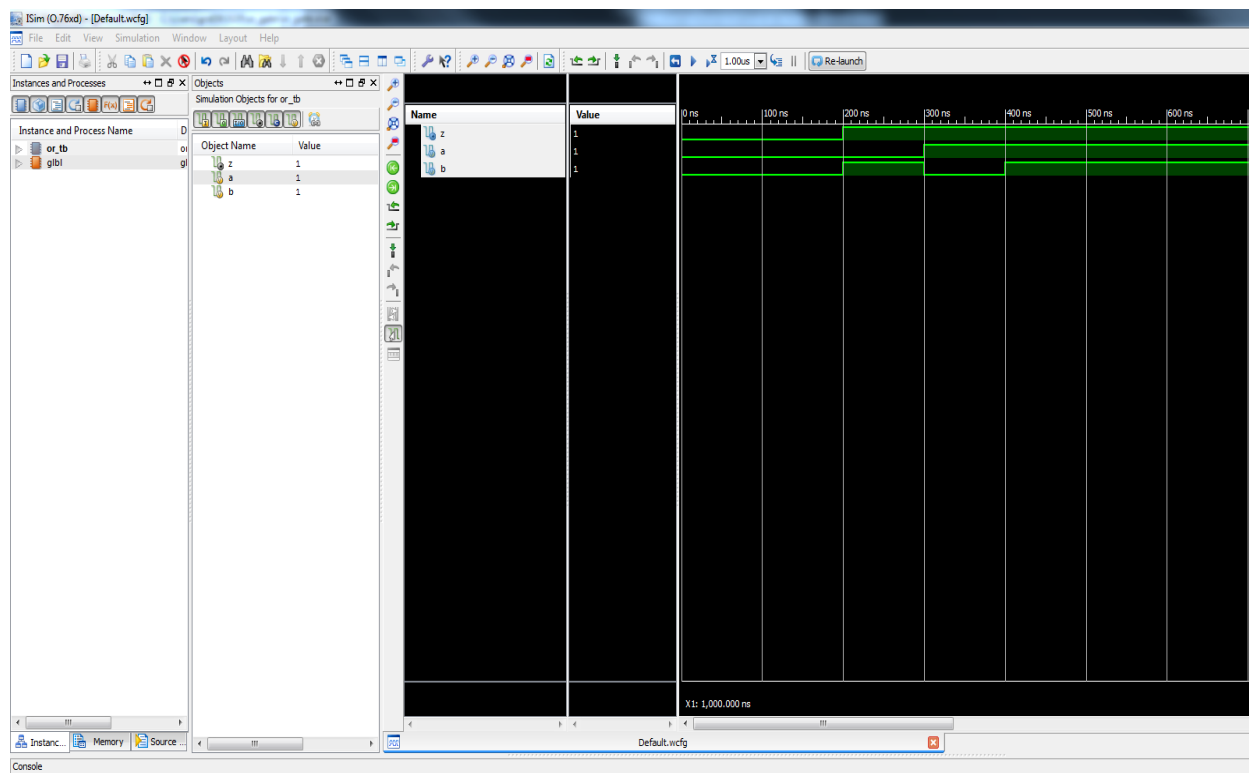


Figure 18 : The iSIM simulator window- behaviorial simulation for or gate design

6. Preparing and downloading bitstream file for the Spartan FPGA:

A bitstream file needs to be prepared for each design and downloaded onto the Digilent prototyping board.

6.1 Generating a User Constraint File

In order to test the design in the Digilent board, the inputs need to be connected to the switches/buttons on the board and the outputs need to be connected to the onboard LED's. This is specified by a “**User Constraint File** (ucf file). To create a UCF for your design do the following:

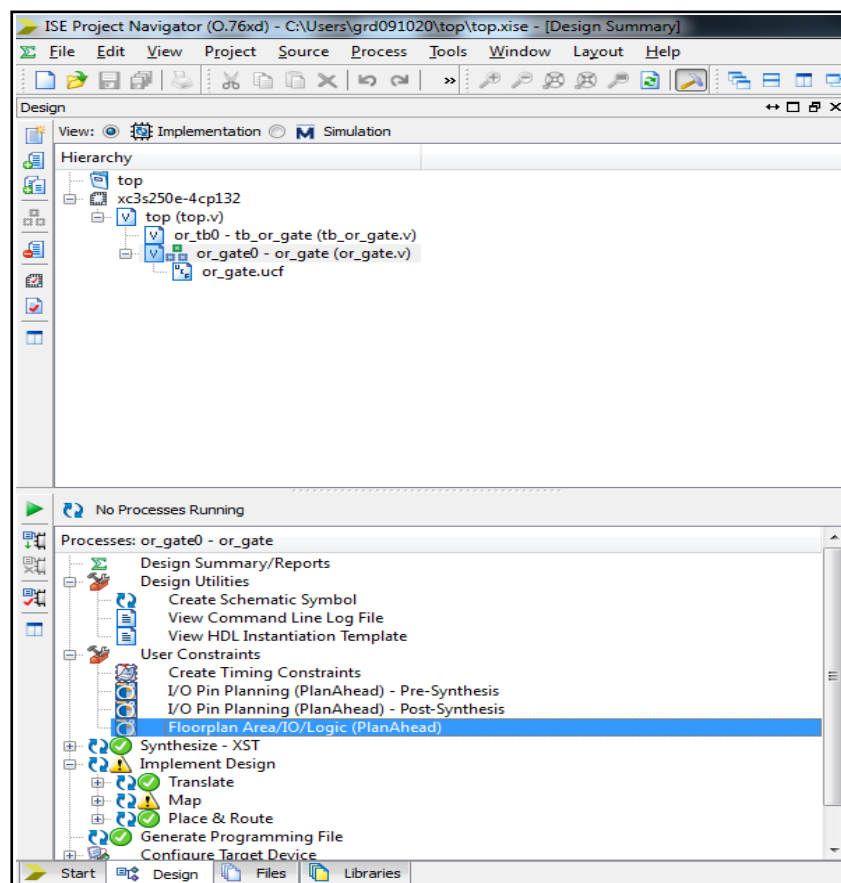


Figure 19: Launching XilinxPlanAhead from Xilinx ISE

Double click on the **Floorplan Area/IO/Logic(Plan Ahead)** option in the process window. This will launch the **Xilinx Plan Ahead** tool.

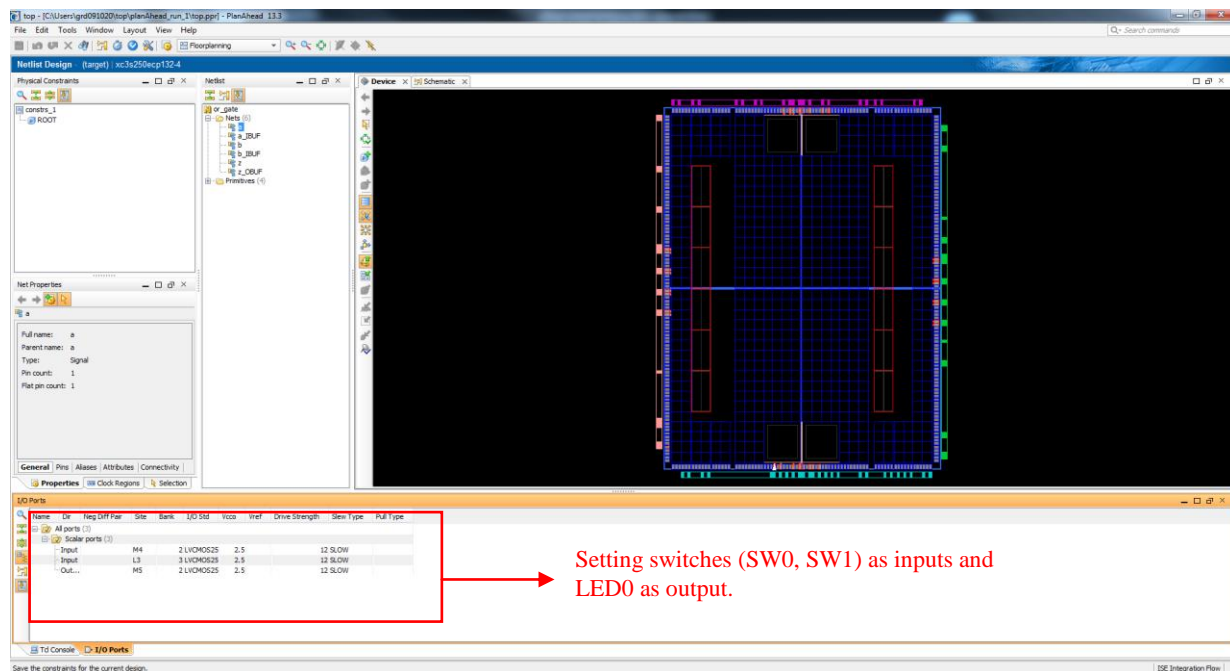


Figure 20: Assigning pins to inputs/outputs in Xilinx PlanAhead.

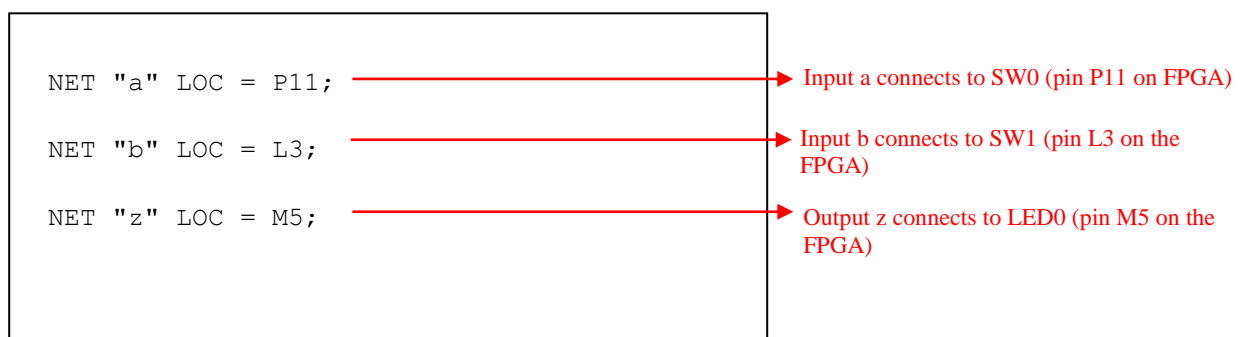
In the **PlanAhead** window shown in [Figure 14](#), click on the nets that are assigned to the design (in this case *a*, *b*, *z*) and in the I/O ports window assign the pin number aliases as specified on the Digilent board.

Refer to I/O pin mapping information for the Digilent board.

- For the OR_GATE example, the user constraints may be specified as follows:

We can use switches SW0 and SW1 on the Digilent board as inputs (*a* and *b*) and LED0 as the output (*z*). Note that:

Pin **P11** and **L3** are FPGA pins connected to SW0 and SW1 on the Digilent Board
 Pin **M5** of the FPGA is connected to **LED0** on the Digilent Board.



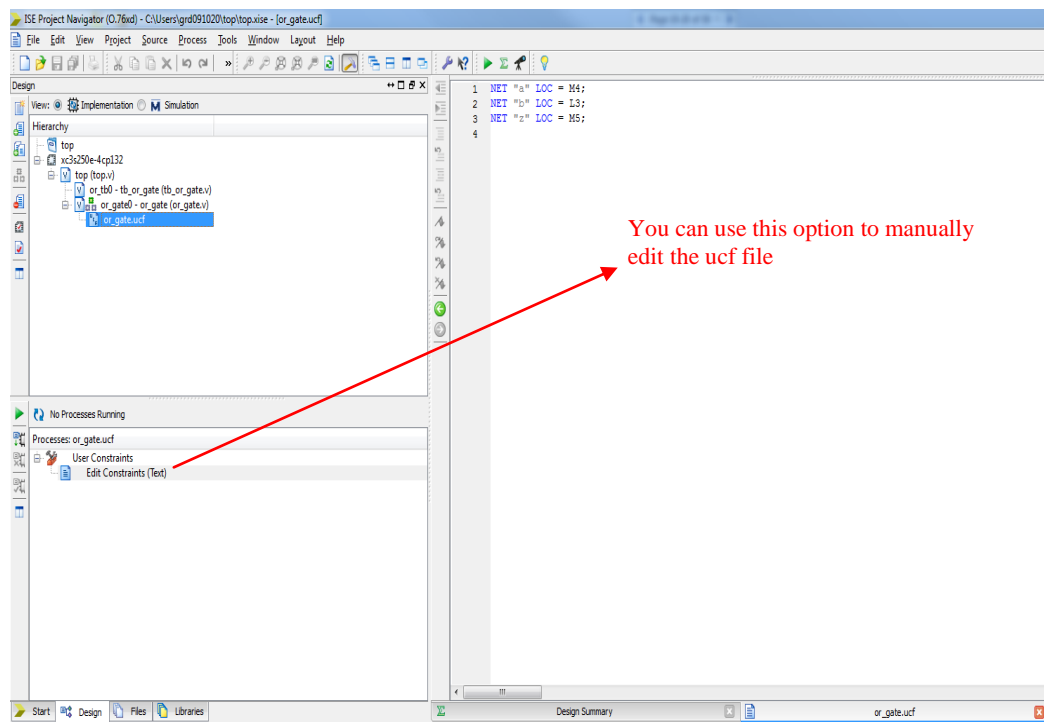


Figure 21: User Constraint File (snapshot from Xilinx ISE software)

You may verify the generated UCF file by clicking on `or_gate.ucf` in the Design window in ISE.

If you would like to modify the UCF file manually (to change the switches etc.) you may do so by clicking on the **Edit Constraints(Text)** option in the Process window ([Figure 15](#))

6.2 Generating a Bit Stream file (.bit)

In order to program the FPGA, you will need to generate a bit stream file of your design. In the Xilinx ISE process window:

- a. Right click on **Generate Programming File** and select Process Properties.

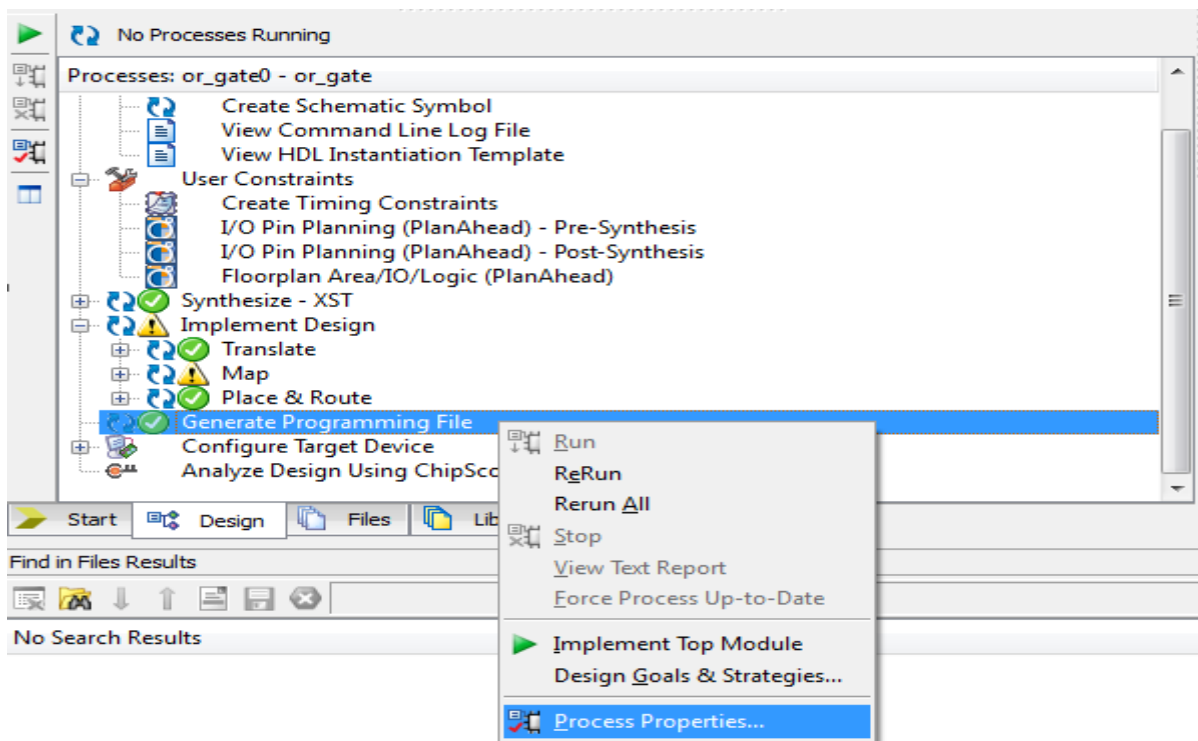


Figure 22: Setting the Process Properties in Xilinx ISE.

- b. In ‘Startup options’ select **JTAG Clock** and click “Apply” and then “OK”.

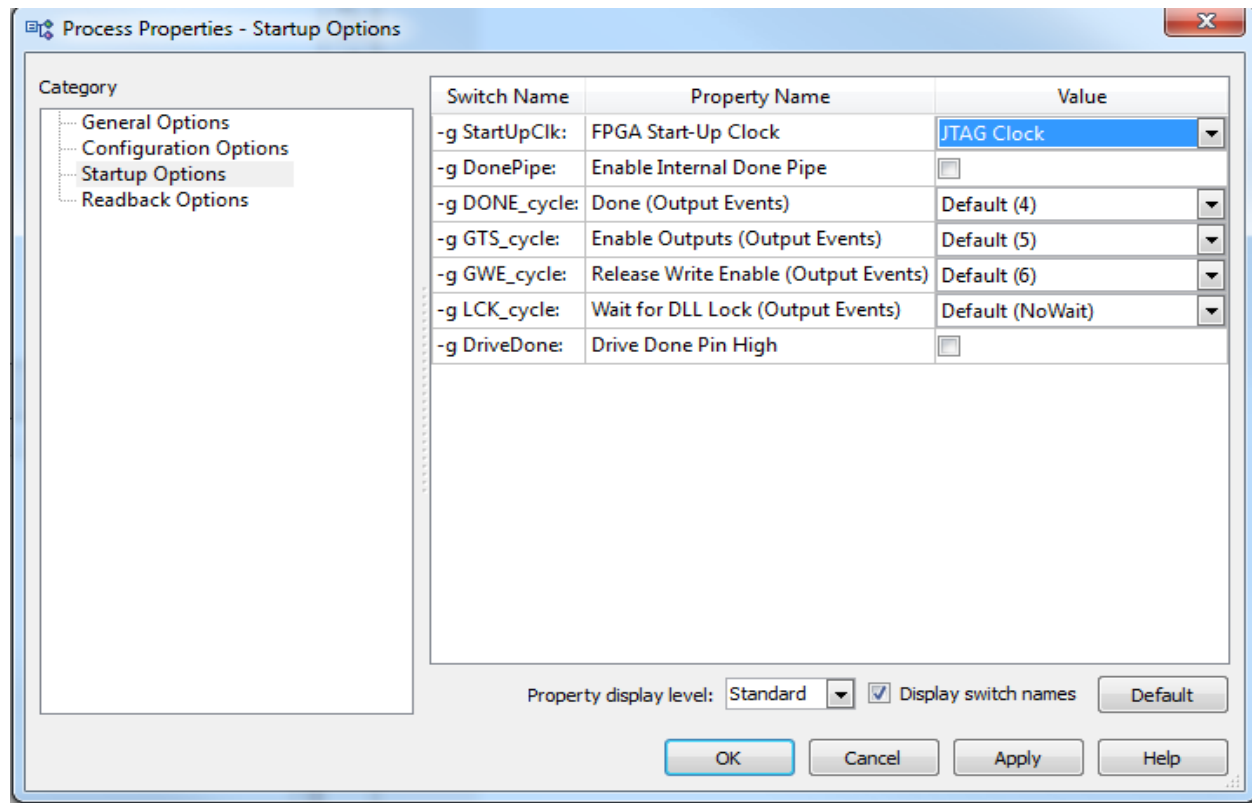


Figure 23: Setting CLK option on Xilinx ISE.

- c. Double click on '**Generate Programming file**' (or right click and choose Run).

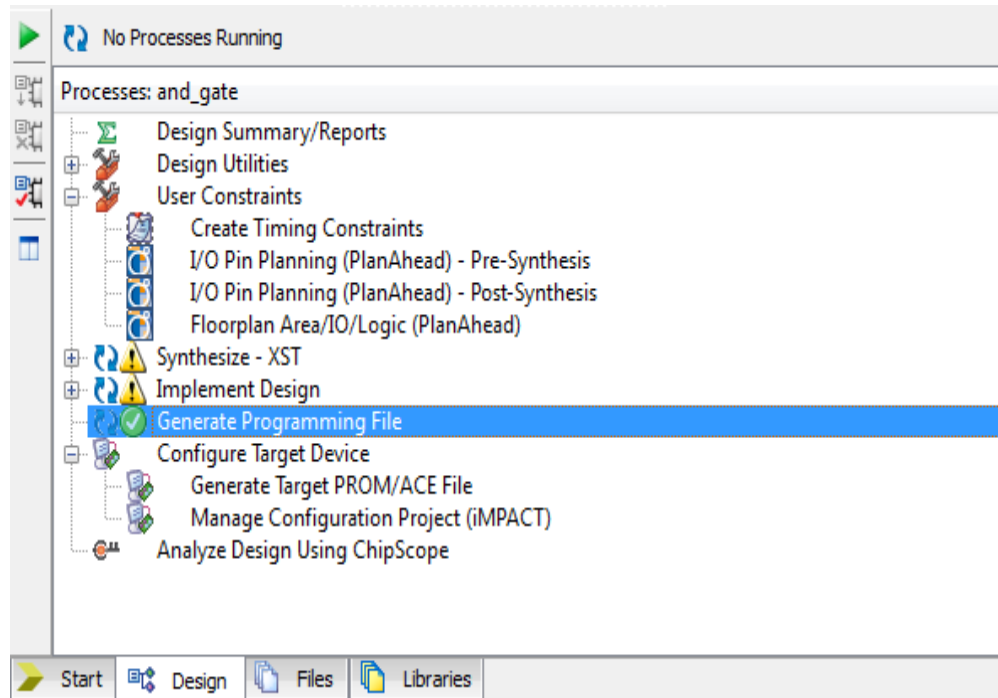


Figure 24: Generating a bistream file for your design.

This will generate a bitstream file that you can use to program the FPGA.

6.2 Programming the Digilent Board using Adept:

To program the FPGA on the Digilent board, do the following:

- Connect the Digilent board to the USB cable (make sure the switch is set to 'ON' position).
- Launch the [Digilent Adept](#) tool (Start->All Programs->Digilent->Adept)

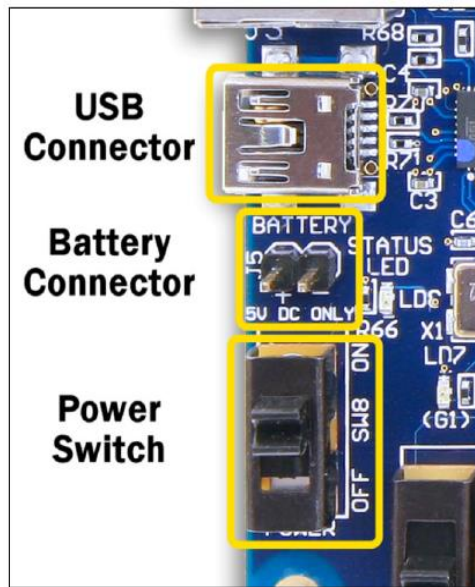


Figure 25: The USB connector and Power switch on the Digilent BASYS board.

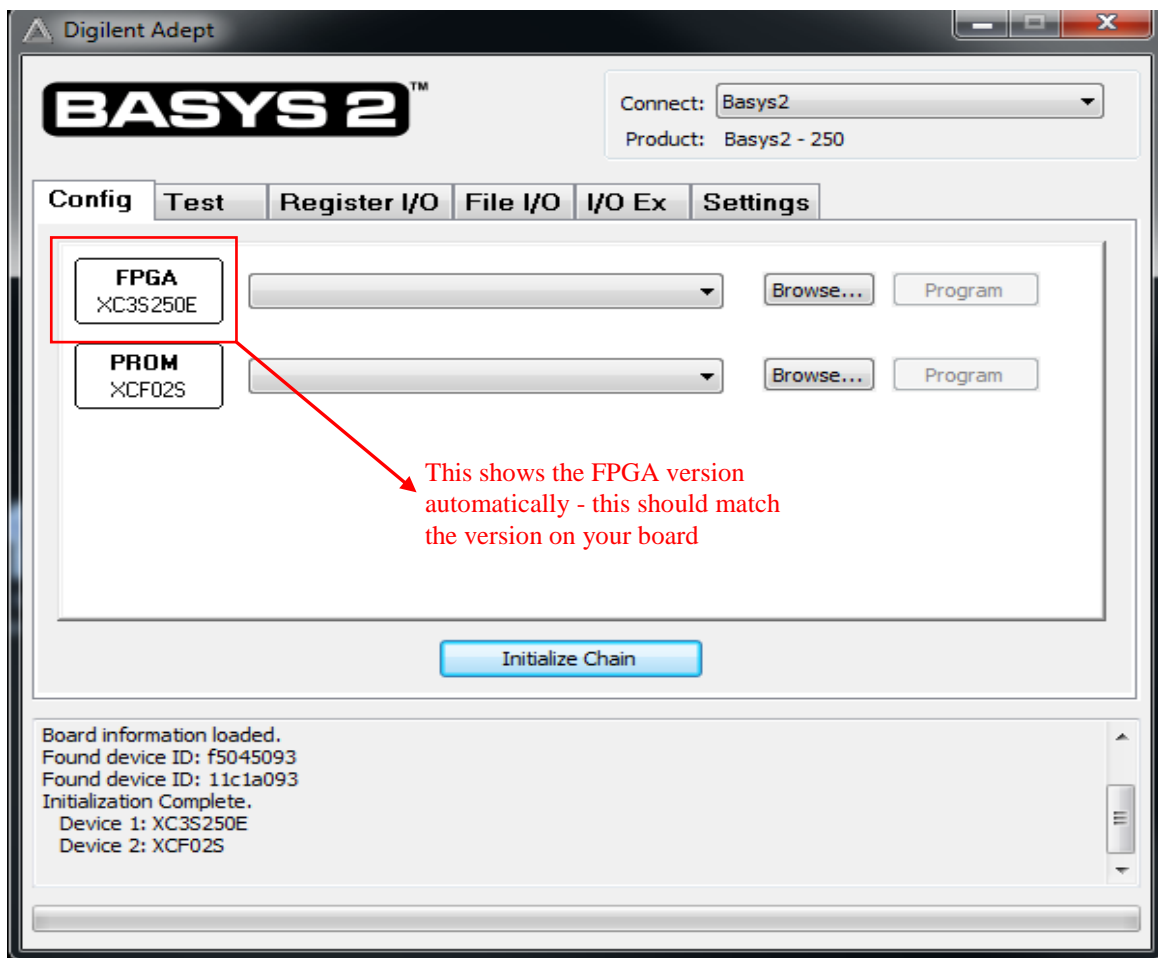


Figure 26: The Digilent Adept window

- Click on Browse and select the bitfile that you generated in step 6.1 (or_gate.bit) and click on **“Program”**

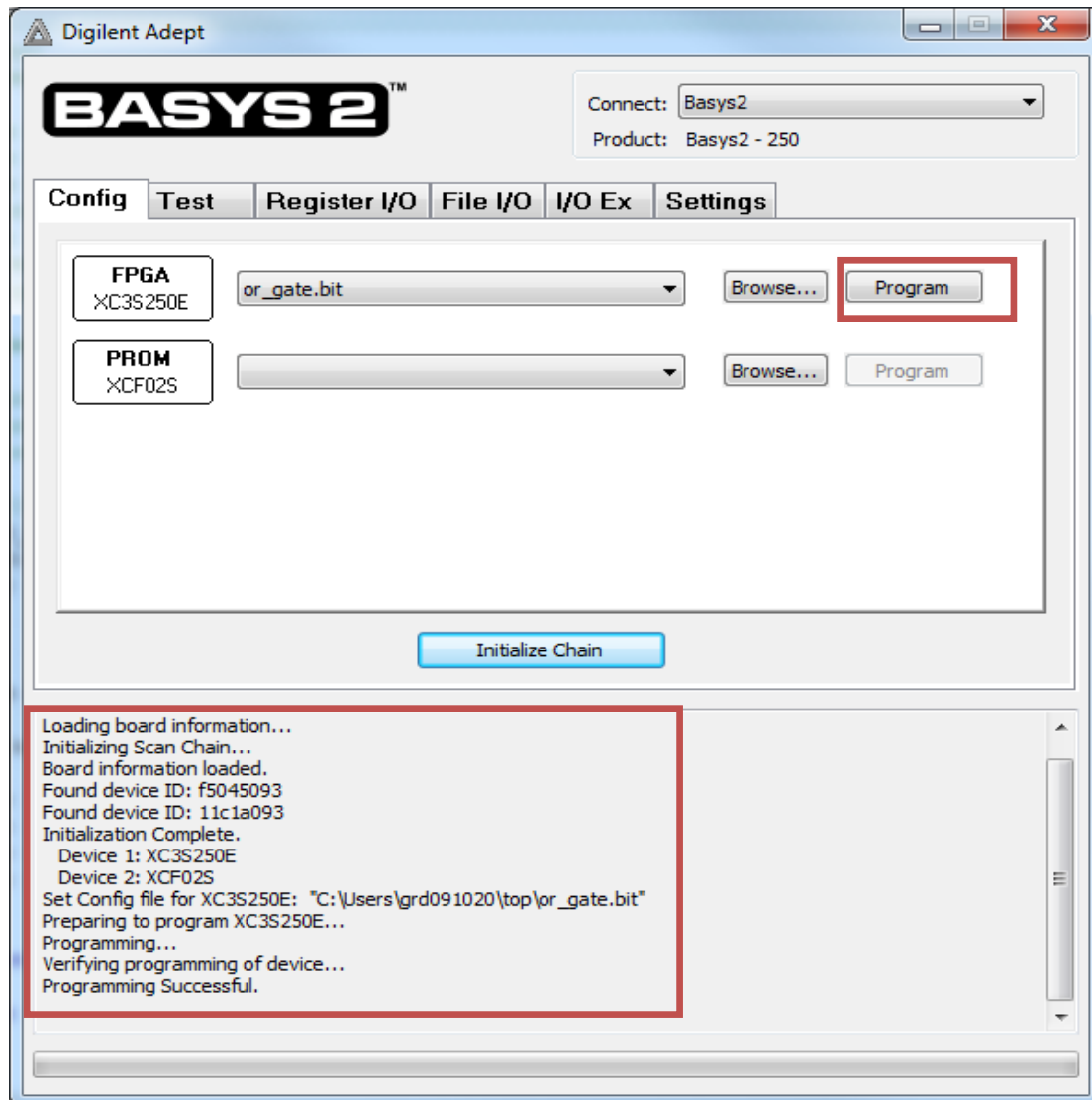


Figure 27: Programming the FPGA using Adept

Now you can test the functional behavior of your design on the FPGA using the switches and LEDs on the BASYS board. Ensure that the **“Programming Successful”** message appears in the message window. If you face problems please ensure:

- The USB cable is connected to the board.
- The board is powered ON. Check the switch settings as shown in [Figure 19](#).
- The FPGA device **on the board matches the one that you specified in your design** properties. Eg. XC3S250E or XC3S100E. Note that if there is a mismatch you will usually encounter an error stating **“Unable to associate file with device due to IDCODE conflict”**. If this occurs, re check your design properties.

7. Testing a Digital Logic Circuit (using the Digilent BASYS board)

Testing a downloaded design requires connecting the inputs of the design to switches or ports and the outputs of the design to LEDs or 7-segment displays. In case of sequential circuits, the clock input(s) must also be connected to clock sources. These inputs and outputs can be connected to appropriately on the Digital Lab workbench.

The **Digilent BASYS board** used in the Digital Circuits lab has the following features, which can be used to test the digital logic in the design:

1. 8 Switches -- which can be used to drive up to 8 inputs
2. 4 Buttons -- which can be used as reset signals (or) input switches
3. 8 LEDs -- which can be used to display up to 8 design outputs
4. 4 Seven segment displays -- which can be used to display four digits of information on the board
5. Mini USB interface
6. Power on switch

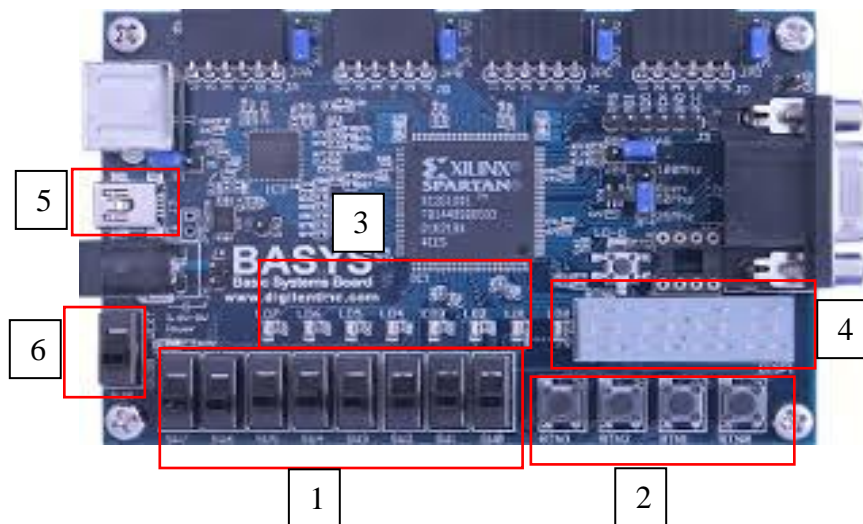


Figure 28: The Digilent BASYS board with Spartan 3 FPGA

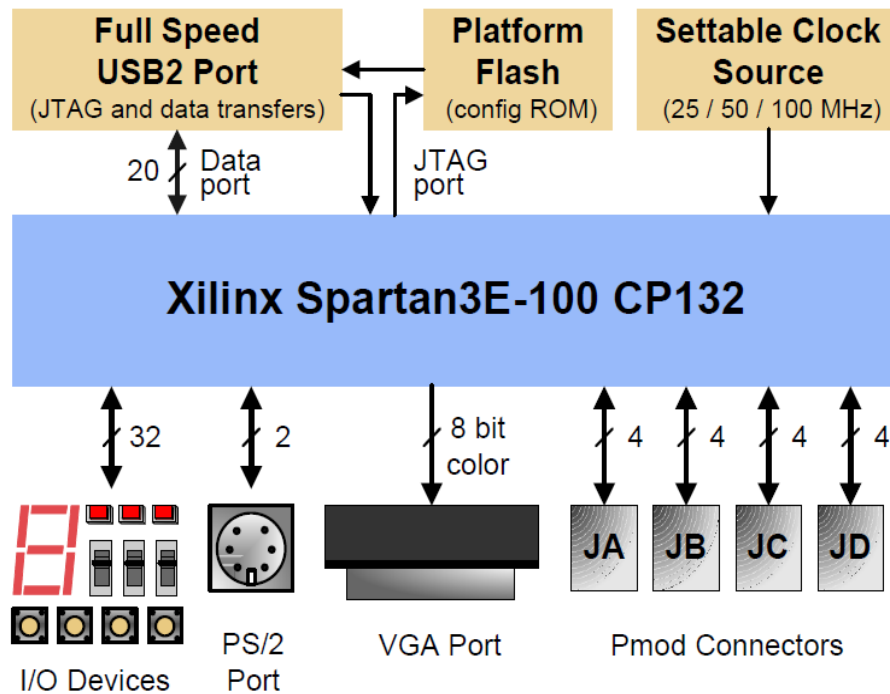


Figure 29: Digilent BASYS Board Architecture (www.digilentinc.com)

In order to use the respective input/output device on the board, the pin number of the device must be connected properly to the design's input/output. On the Digilent board, the pin number of these inputs/outputs is as follows:

	Input	FPGA Pin (To be used in the ucf file)
Switches	SW0	pin# M4
	SW1	pin# L3
	SW2	pin# K3
	SW3	pin# B4
	SW4	pin# G3
	SW5	pin# F3
	SW6	pin# E2
	SW7	pin# N3
Button	BTN0	pin# G12
	BTN1	pin# C11
	BTN2	pin# M4
	BTN3	pin# A7

	Output	FPGA Pin (To be used in the ucf file)
LED	LD0	pin# M5
	LD1	pin# M11
	LD2	pin# P7
	LD3	pin# P6
	LD4	pin# N5
	LD5	pin# N4
	LD6	pin# P4
	LD7	pin# G1

Table 1: Pin mapping on the Digilent BASYS board

7.1 Observing outputs using the on-board LEDs and Seven Segment Displays

The **Digilent** board has four on-board 7-segment displays (see [Figure 22](#)) that are connected to the corresponding on-board Spartan 3E FPGA chip. This display can be used to observe the outputs of your design without using any additional wires if the design conforms to the pin assignments for the on-board 7-segment display. Figure 22 shows the 7-segment display with the conventional labeling of individual segments.

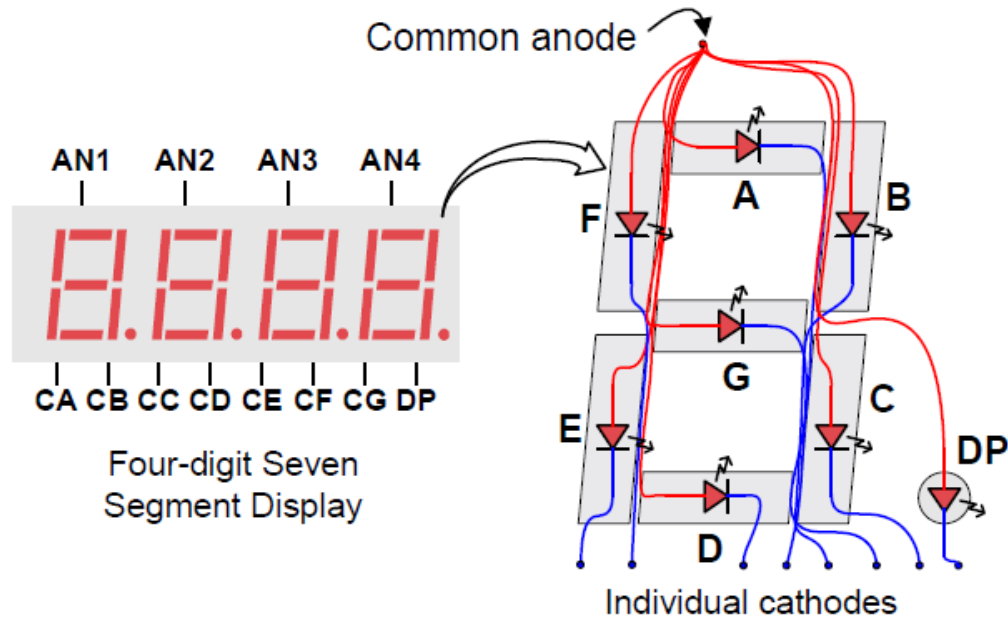


Figure 30: 7-segment display (source: Basys user's manual)

The Digilent board contains a 4-digit common anode seven-segment LED display. The display is multiplexed, so only seven cathode signals (CA,CB,CC,CD,CE,CF,CG) exist to drive all 28 segments in the display. Four digit-enable signals (AN0, AN1, AN2, AN3) drive the common anodes, as shown in [Figure 22](#) and these signals determine which digit the cathode signals illuminate.

	Output	FPGA Pin (To be used in the ucf file)
LED Anode (To be used to Multiplex between Four Displays)	AN0	pin# F12
	AN1	pin# J12
	AN2	pin# M13
	AN3	pin# K14
LED Cathode	CA	pin# L14
	CB	pin# H12
	CC	pin# N14
	CD	pin# N11
	CE	pin# P12
	CF	pin# L13
	CG	pin# M12

Table 2: Seven segment display LED mapping on the Digilent board

This connection scheme creates a multiplexed display, where driving the anode signals and corresponding cathode patterns of each digit in a repeating, continuous succession can create the appearance of a 4-digit display. Each of the four digits will appear bright and continuously illuminated if the digit enable signals are driven low once every 1 to 16ms (for a refresh frequency of 1KHz to 60Hz).

The Seven segment display timing to drive all the four displays is shown below:

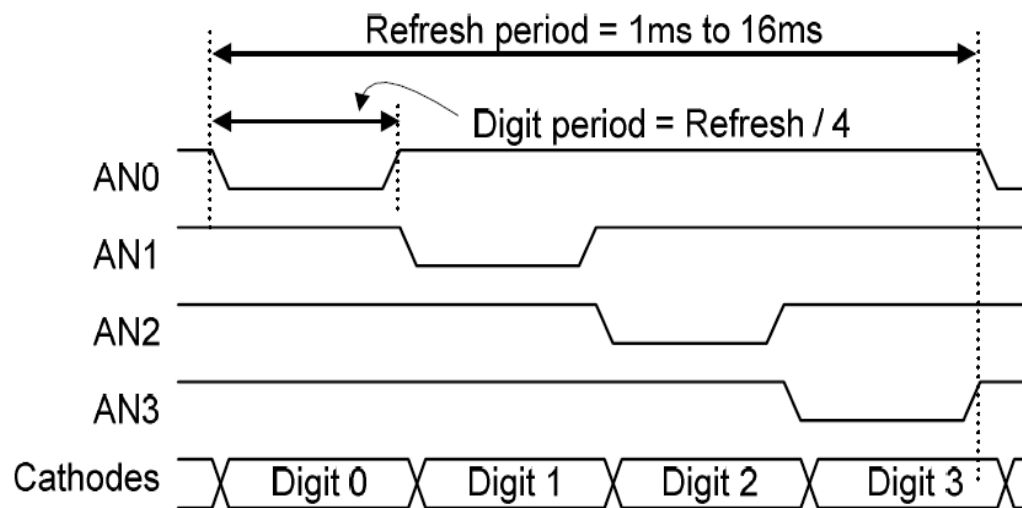


Figure 31: Timing diagram for Multiplexed Seven Segment Displays

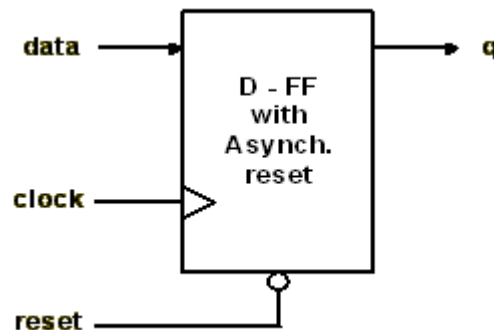
8. Design and Simulation of Sequential Circuits using Verilog HDL

The procedure to create Verilog design files for sequential circuits in Xilinx ISE is the same as that for combinational circuits. The main difference between combinational and sequential designs is the presence of flip-flops (**registered** outputs or nodes in the Declaration section of a sequential design).

8.1 Design of Sequential Circuits

For large, complex state machines it is easier to specify them as programs. A sequential circuit can be described either as a procedural block or a state machine in Verilog.

1. A D-flip with asynchronous reset can be modeled as a Procedural block as follows:



```
module dff_async (data, clock, reset, q);
    input  data, clock, reset;
    output q;

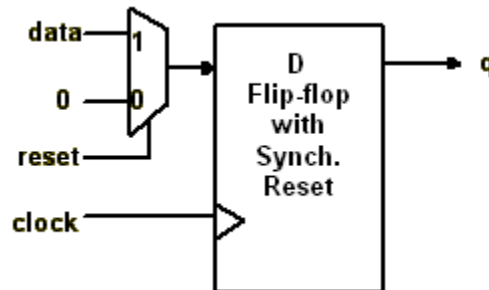
    reg    q;

    // logic begins here

    always @(posedge clock or reset)

        if(reset == 1'b0)
            q <= 1'b0;
        else
            q <= data;
endmodule
```

2. A D-flip with synchronous reset can be modeled as a Procedural block as follows:



```
module dff_sync (data, clock, reset, q);
    input  data, clock, reset;
    output q;

    reg    q;

    // logic begins here
    always @(posedge clock)
        if(reset == 1'b0)
            q <= 1'b0;
        else
            q <= data;
endmodule
```

8.2 Simulation of sequential designs

Except for the additional clock signal, simulation of sequential designs can be done using test_bench in the same way it was done for combinatorial circuits. The clock signal can be generated in the test bench using a simple initial block as follows:

```
module test_bench(clk)
    output clk;
    reg clk;

    initial begin
        clk = 0;
        forever begin
            #5 clk = ~clk; //Time period of the clock is 10 time units.
        end
    end

    // rest of the logic

endmodule
```

9. Design and Simulation of Finite State machines in Verilog.

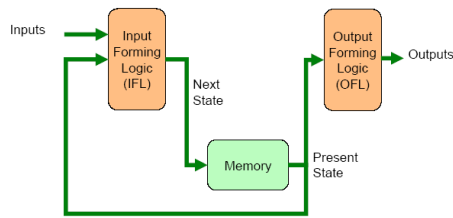
Basically a Finite State Machine (FSM) consists of a combinational logic, sequential logic and output logic. Where combinational logic is used to decide the next state of the FSM, sequential logic is used to store the current state of the FSM.

Types of State Machines There are many ways to code finite state machines, but before we get into the coding styles, it is important to understand the basics.

There are two types of state machines, Mealy and Moore:

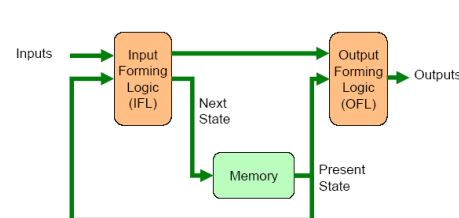
Sequential Logic – Moore Model

- In a Moore model outputs of a circuit are function of the present state only.



Sequential Logic – Mealy Model

- In a Mealy model outputs of a circuit are a function of the present state and the present inputs of the circuit.



Source: EE3320 – Digital Circuits course slides

Depending on the need, either type of state machine can be used.

Encoding Style Since the state machine needs to be represented as a digital circuit, the states can be represented by the following ways:

- **Binary encoding** : In this each of the state is represented in binary code (i.e. 000, 001, 010....)
- **Gray encoding** : In this each of the state is represented in gray code (i.e 000, 001, 011,...)
- **One Hot** : In this only one bit is high and rest are low(i.e. 0001, 0010, 0100, 1000)
- **One Cold** : In this only one bit is low, rest are high (i.e. 1110,1101,1011,0111)

9.1 State Machine Design in Verilog

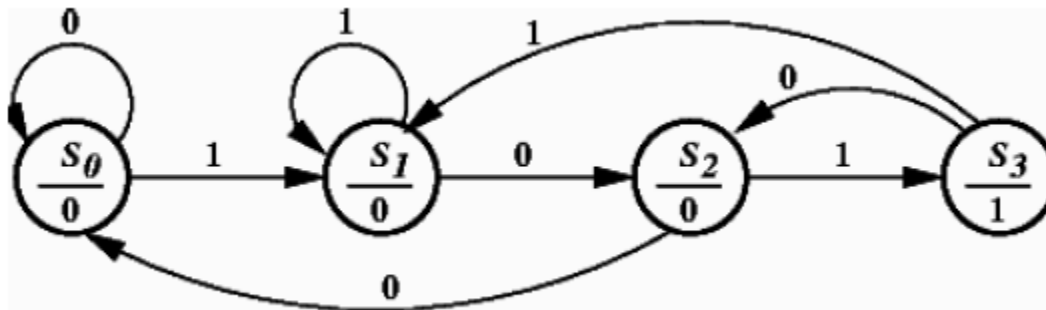
State machine design in Verilog is illustrated with the following example.

Example: Use Verilog HDL to design a sequence detector with one input X and one output Z. The detector should recognize the input sequence “101”. The detector should keep checking for the appropriate sequence and should not reset to the initial state after it has recognized the sequence. The detector initializes to a reset state when input, **RESET** is activated.

For example, for input X = “...110110101...”,
FSM output Z = “...000100101...”

9.1.1 Moore State Machine Example:

The state machine diagram of the Moore State machine for sequence detector example is shown below:



```
`define CK2Q 5 // Defines the Clock-to-Q Delay of the flip-flop
module moore_fsm(reset,clk,in_seq,out_seq);
input reset;
input clk;
input in_seq;
output out_seq;
reg out_seq;
//-----Parameters defining State machine States-----
parameter SIZE = 4;
parameter S0 = 4'b0001 , S1 = 4'b0010 , S2 = 4'b0100, S3 = 4'b1000;

//-----Internal Variables-----
reg [SIZE-1:0] state ;// Seq part of the FSM
reg [SIZE-1:0] next_state ;// combo part of FSM

//-----Moore State machine Code starts Here-----
// Determine the next state for each state in the state machine using the input
// sequence given to it. The output next_state is combinatorial in nature.
//-----

always @ (state or in_seq)
begin : FSM_COMBO

next_state = 4'b0001;

case(state)
S0 : if (in_seq == 1'b1) begin
next_state = S1;
end
else begin
next_state = S0;
end

S1 : if (in_seq == 1'b0) begin
next_state = S2;
end
else begin
next_state = S1;
end

S2 : if (in_seq == 1'b1) begin
next_state = S3;
end
else begin
next_state = S0;
end

S3 : if (in_seq == 1'b1) begin
next_state = S1;
end
else begin
next_state = S2;
end

end
```

```

end

    // Always include a Default state in your state machine.
    default : next_state = S0;
endcase
end

//-----
// Register the combinatorial next_state output.
//-----
always @ (posedge clk)
begin : FSM_SEQ
    if (reset == 1'b0) begin
        state <= #`CK2Q S0;
    end
    else begin
        state <= #`CK2Q next_state;
    end
end

//-----
// Based on the current state only, determine the output "out_seq"
//-----

always @ (state or reset)
begin : OUTPUT_LOGIC
    if (reset == 1'b0) begin
        out_seq <= #`CK2Q 1'b0;
    end
    else begin
        case(state)
            S0 : begin
                out_seq <= 1'b0;
            end
            S1 : begin
                out_seq <= 1'b0;
            end
            S2 : begin
                out_seq <= 1'b0;
            end
            S3 : begin
                out_seq <= 1'b1;
            end
            default : begin
                out_seq <= 1'b0;
            end
        endcase
    end
end

end // End Of Block OUTPUT_LOGIC
endmodule // End of Module Moore state machine
//-----

```

Except for the additional clock signal, simulation of finite state machines can be done using a test_bench in the same way it was done for combinatorial circuits. The following is a sample test-bench for the Moore state machine. The same can be used to test the Mealy state machine as well.

```

//-----
module sm_tb(reset,clk,in_seq,out_seq);
    output reset;
    output clk;
    output in_seq;
    input out_seq;

    reg clk;
    reg reset;
    reg [15:0] data;
    reg in_seq;
    integer i;

```

```

// The input data sequence is defined in the vector "data"
// Each clock one bit of data is sent to the state machine
// which will detect the sequence "101" in this data.
initial
begin
    data = 16'b0010100110101010;
    i = 0;
    reset = 1'b0;
    #1200 ;
    reset = 1'b1;
    #60000;
    $finish;
end

// Clock Generation
initial
begin
    clk = 0;
    forever begin
        #600 ;
        clk = ~clk;
    end
end

// Right shifting of data to generate the input sequence
always @(posedge clk)
begin
    #50;
    in_seq = data >> i;
    i = i+1;
end

```

endmodule

//-----

A top level module is required to stitch the state machine to the test bench.

//-----

```

module top();
wire clk,reset;
wire in_seq,out_seq;

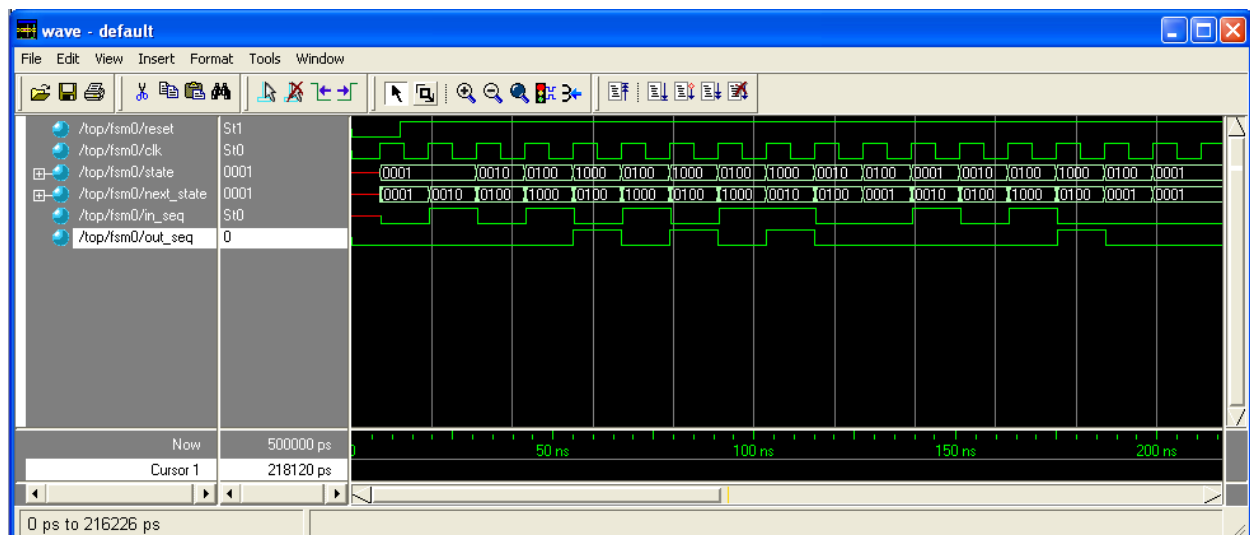
    moore_fsm fsm0(reset,clk,in_seq,out_seq);
    sm_tb fsm_tb(reset,clk,in_seq,out_seq);

```

endmodule

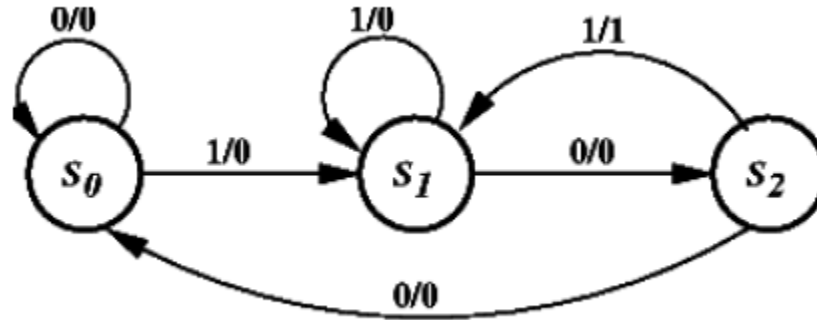
//-----

The operation of the Moore state machine is shown in the waveform below:



9.1.2. Mealy State Machine Example:

The state machine diagram of the Mealy State machine for sequence detector example is shown below:



```
`define CK2Q 5 // Defines the Clock-to-Q Delay of the flip flop.

module mealy_fsm(reset,clk,in_seq,out_seq);
input reset;
input clk;
input in_seq;
output out_seq;

reg out_seq;
reg in_seq_reg;

//-----Parameters defining State machine States-----
parameter SIZE = 3;
parameter S0 = 3'b001 , S1 = 3'b010 , S2 = 3'b100;

//-----Internal Variables-----
reg [SIZE-1:0] state ;// Seq part of the FSM
reg [SIZE-1:0] next_state ;// combo part of FSM

//-----Register the input-----
always @ (posedge clk)
begin : REG_INPUT
    if (reset == 1'b0) begin
        in_seq_reg <= #`CK2Q 1'b0;
    end
    else begin
        in_seq_reg <= #`CK2Q in_seq;
    end
end

//-----Mealy State machine Code starts Here-----
// Determine the next state for each state in the state machine using the input
// sequence given to it. 'next_state' is combinatorial in nature.
//-----
always @ (state or in_seq_reg)
begin : FSM_COMBO

    next_state = 3'b000;

    case(state)
        S0 : if (in_seq_reg == 1'b1) begin
            next_state = S1;
        end
        else begin
            next_state = S0;
        end
    end
end
```



```

S1 : if (in_seq_reg == 1'b0) begin
    next_state = S2;
end
else begin
    next_state = S1;
end

S2 : if (in_seq_reg == 1'b1) begin
    next_state = S1;
end
else begin
    next_state = S0;
end

default : next_state = S0;
endcase
end

//-----
// Register the combinatorial next_state variable.
//-----
always @ (posedge clk)
begin : FSM_SEQ
    if (reset == 1'b0) begin
        state <= #`CK2Q S0;
    end
    else begin
        state <= #`CK2Q next_state;
    end
end

//-----
// Based on the combinatorial next_state signal and the input sequence, determine the output
// out_seq of the finite state machine.
//-----
always @ (state or in_seq_reg or reset)
begin : OUTPUT_LOGIC
    if (reset == 1'b0) begin
        out_seq <= 1'b0;
    end
    else begin
        case(state)
        S0 : begin
            out_seq <= 1'b0;
        end
        S1 : begin
            out_seq <= 1'b0;
        end
        S2 : begin
            if (in_seq_reg == 1'b1)
                out_seq <= 1'b1;
            else
                out_seq <= 1'b0;
            end
        default : begin
            out_seq <= 1'b0;
        end
        endcase
    end
end

end // End Of Block OUTPUT_LOGIC
endmodule // End of Module Mealy state machine

```

Except for the additional clock signal, simulation of finite state machines can be done using a test_bench in the same way it was done for combinatorial circuits. The test bench used to test the Moore State machine can be used to test the Mealy state machine as well.

The screenshot shows the Xilinx ISE Waveform Editor interface. The title bar indicates the project is 'wave - default'. The menu bar includes File, Edit, View, Insert, Format, Tools, and Window. The toolbar contains various icons for file operations, editing, and viewing. The waveform area displays a timing diagram for several signals. The signals listed in the left pane are: /top/ism0/reset (SI1), /top/ism0/clk (SI0), /top/ism0/state (O01), /top/ism0/next_state (O01), /top/ism0/in_seq (SI0), /top/ism0/in_seq_reg (0), and /top/ism0/out_seq (0). The waveform shows a sequence of events over time, with a time scale of 50 ns. The cursor is positioned at 326010 ps. The bottom status bar shows the current time as 0 ps to 245898 ps.

The following factors are taken into account while designing a state machine:

- ptb/dkb 2013*

10. Hierarchical Circuit Design Using Modules

It is always a good practice to keep a design modular and hierarchical. This is important for designs of moderate to high complexity. [Refer to section on hierarchies and Instantiation in the Verilog tutorial in Appendix-A]. Often, you will use a circuit (module) over and over again. Instead of creating these modules every time you need them, it would be more efficient to make a cell or module out of them. You can then use this module every time you need it by instantiating the module in your circuit. Verilog supports hierarchical design by creating *instances* of another modules that can be used in a design. In the example depicted in Figure 24, a 4-bit equivalence circuit is designed using 1-bit equivalence circuit modules.

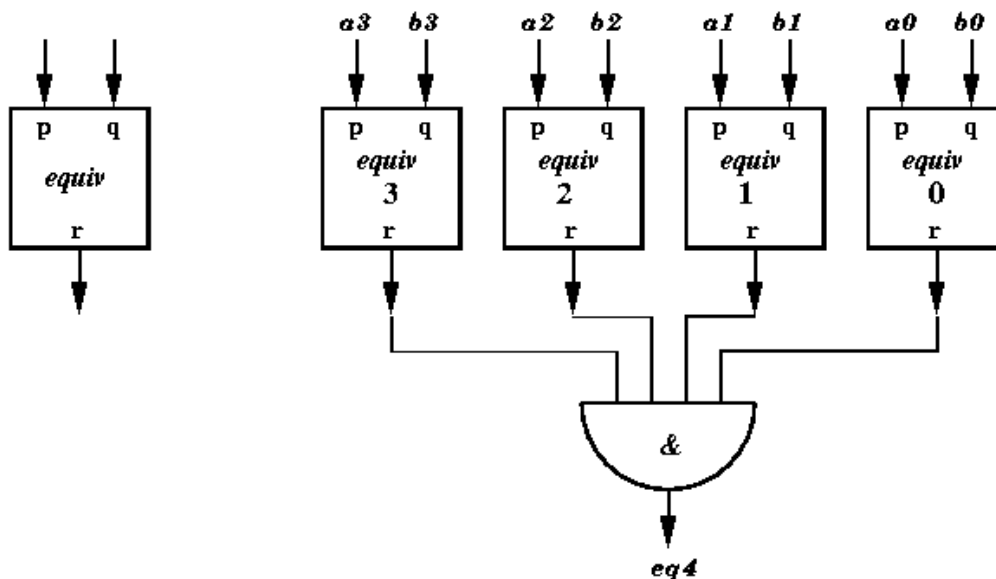


Figure 24: Hierarchical circuit design example: 4-bit equivalence circuit

- **Module Definition:** A module (functional block) definition is specified in a file, separate from the top-level design file using the module.

```
module equiv (p,q,r)
input p;
input q;
output r;

assign r = ~(p ^ q);    //equivalence function is xnor function.

endmodule
```

- **Module Usage:** A design using a module includes a declaration of module interface and instantiation of each module in the Declaration section. Instantiation of module “equiv” in the 4-bit equivalence circuit shown in Figure.21 can be done as follows:

```
module equiv4bit(a3,b3,a2,b2,a1,b1,a0,b0,eq4)
    input a3,b3,a2,b2,a1,b1,a0,b0;
    output eq4;

    equiv eq0(a0,b0,r0);
    equiv eq1(a1,b1,r1);
    equiv eq2(a2,b2,r2);
    equiv eq3(a3,b3,r3);
    assign eq4 = r0 & r1 & r2 & r3;

endmodule
```

NOTE : *For creation of the module, we can either use the design wizard provided by the Xilinx or create our own.*

11. Post Synthesis Timing Simulation

The Xilinx ISE contains a simulator called iSIM which can be used to verify the behavior of your designs. You may also use the Modelsim simulator which can also be invoked directly from Xilinx in order to perform functional and timing simulations. It has an Integrated Development environment that can be used to develop and debug Verilog synthesizable and behavioral logic. Section 5.2 explains usage of the Modelsim simulator from Xilinx ISE to verify the functionality of a given digital logic circuit using a test bench. This chapter deals with performing post synthesis timing simulations using the Modelsim simulator.

We will look at simulation and timing synthesis using an example of a **4X1 multiplexer**:

In order to perform the simulation, we will need two files- a design file and a test bench (as shown below):

The multiplexer design file (mux4x1.v):

```
module mux4x1 (a,b,c,d,sel,y);
    input a, b, c, d;
    input [1:0] sel;
    output y;
    reg y;
    always @ (a or b or c or d or sel)
        case (sel)
            0 : y = a;
            1 : y = b;
            2 : y = c;
            3 : y = d;
            default : y=1'b0; //One bit binary value '0'
        endcase
    endmodule
```

A sample test bench for the multiplexer:

```
module mux4x1_tb ();
    //output a, b, c, d;
    //output [1:0] sel;
    //input y;
    reg a,b,c,d;
    reg [1:0] sel;

    initial
    begin
        a <= 1'b1;
        b <= 1'b0;
        c <= 1'b1;
        d <= 1'b0;

        // Wait 100 ns for global reset to finish
        #100;
        // Add stimulus here
        sel <= 2'b00;
        #200;
        a<=1'b1;
        b<=1'b1;
    end
```

```

    sel <= 2'b01;
    #400;
    sel <= 2'b10;
    #600;
    sel <= 2'b11;
    #800;
    sel <= 2'b00;end

//mux4x1 mux4x1_inst(a,b,c,d,sel,y);
endmodule

```

As explained in the previous section, create a new project named mux4X1 and add a new verilog source (mux4X1.v) with the code shown above.

To add a test bench select the project in the source list and right click to select ‘Add New source’
When the New source wizard pops up, select **Verilog Text Fixture** and choose an appropriate name for your test bench (say, **mux4X1_tb**) and click on **Next**.

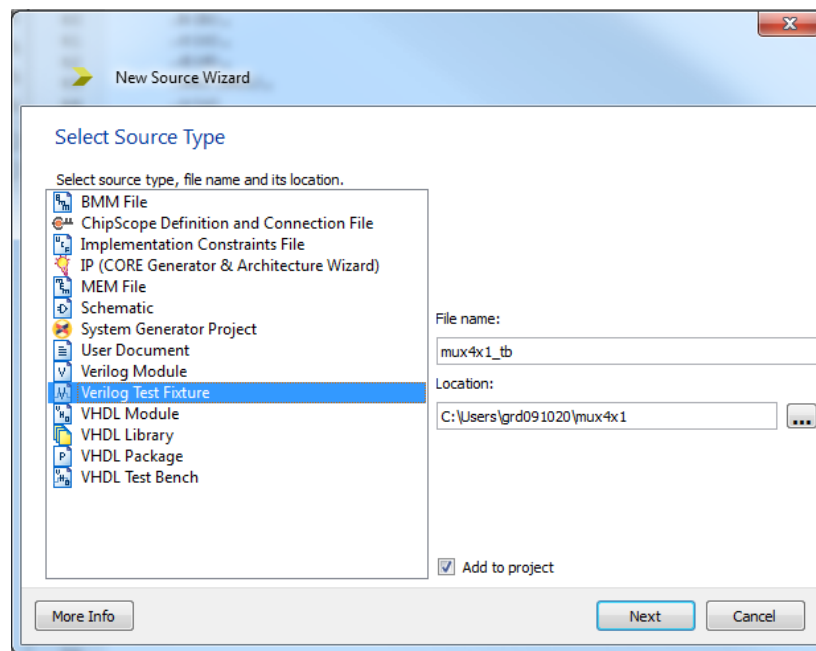


Figure 26: Creating a text fixture in Xilinx ISE

Associate your design with the test bench that you are about to create by selecting mux4x1 as the source and click on **Next** and then **Finish**.

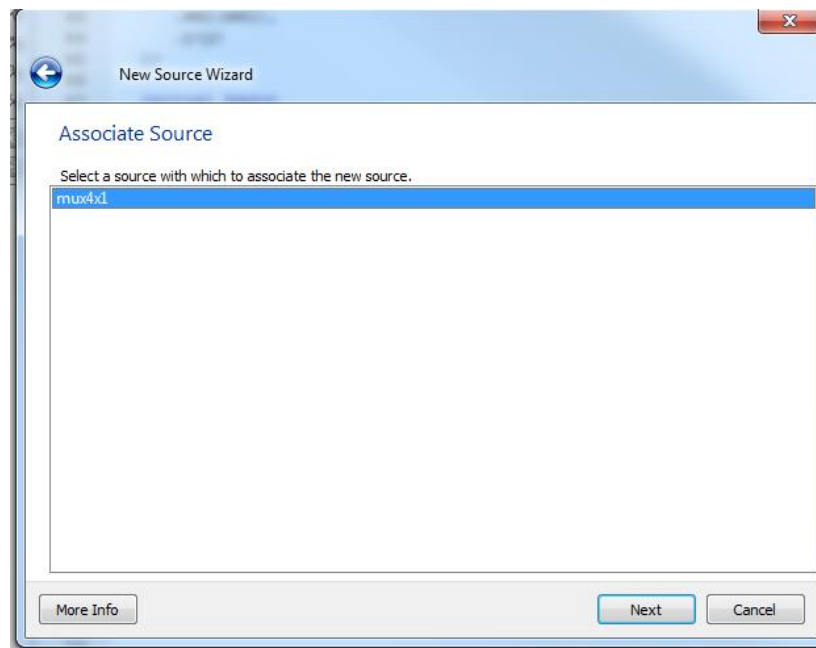


Figure 27: Associating the testbench with the design

The tool generates a test bench with some boilerplate code as shown in Figure

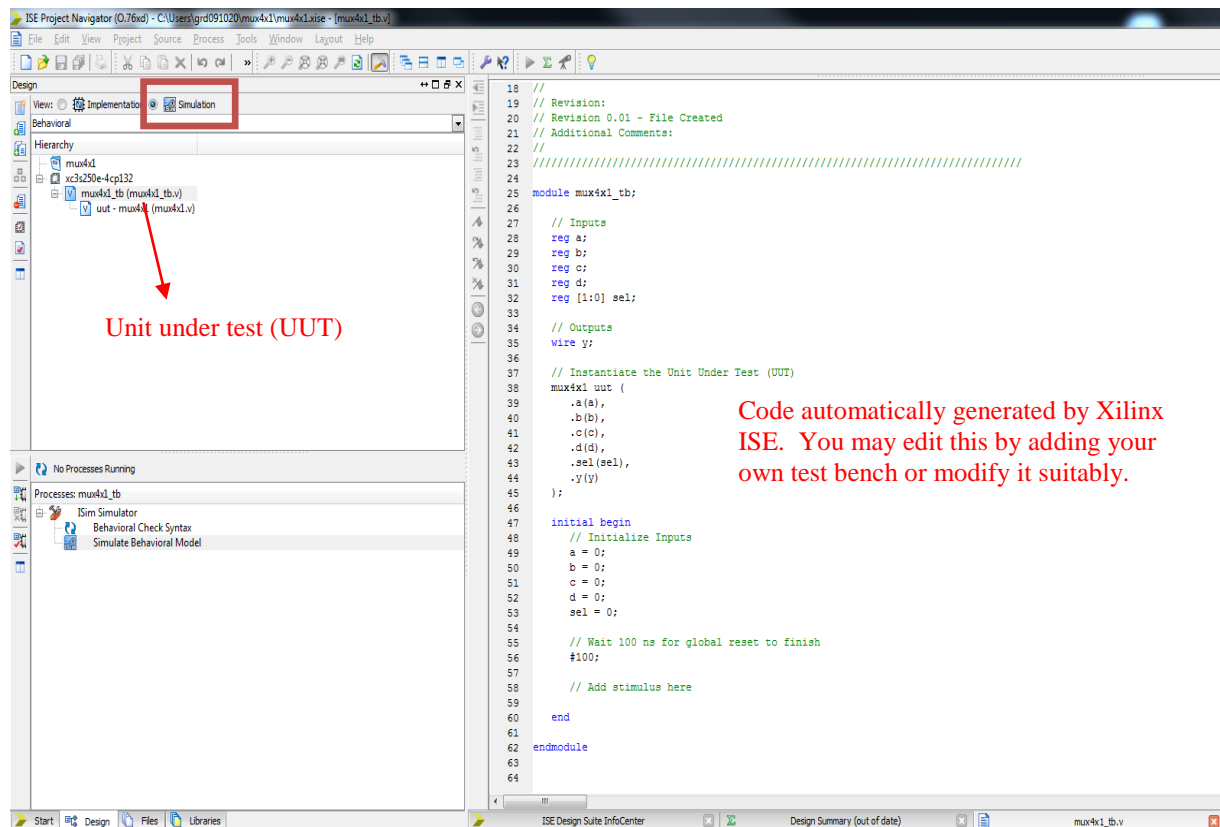



Figure 28: Adding the verilog testbench to the project

Double click on '**Behavioral Check Syntax**' in the process window to make sure there are no syntax errors in your code. A  mark will appear if there are no errors as shown in Figure

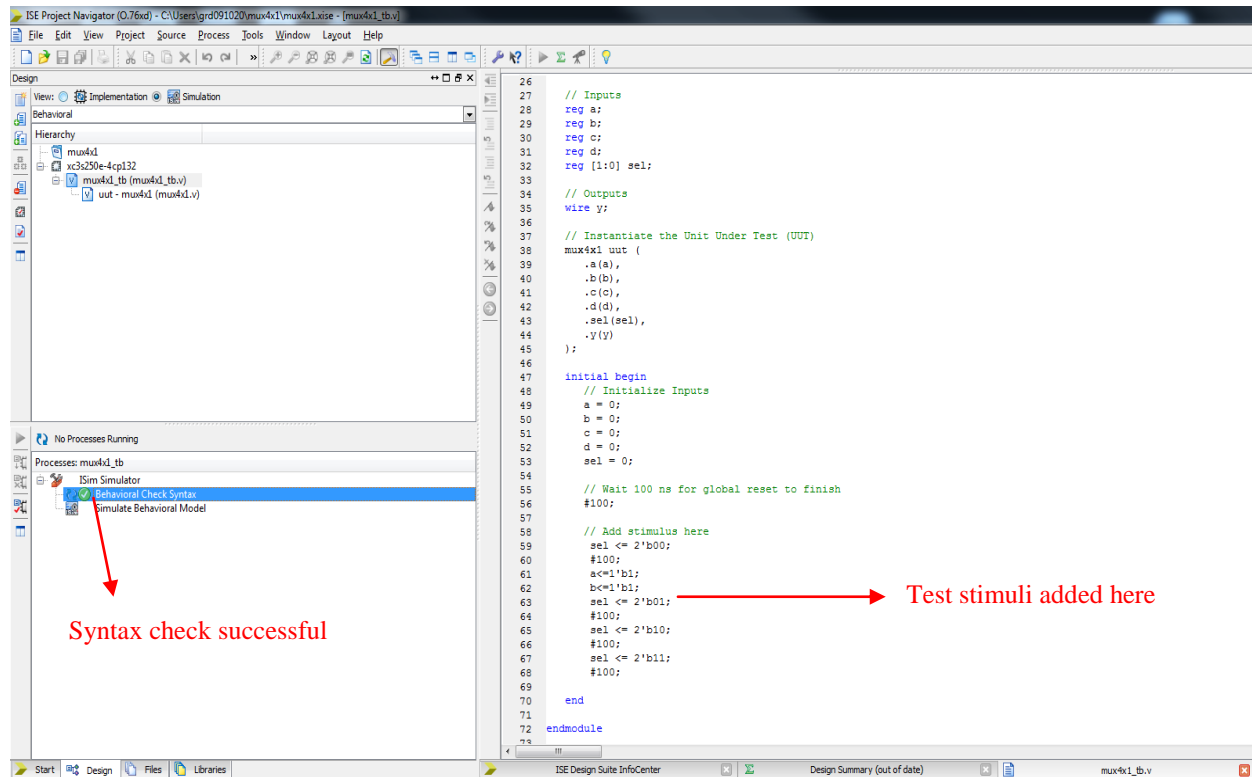


Figure 29: Checking syntax

Now you are ready to simulate your design!

Behavioral simulation using iSIM:

In order to run the iSIM simulator, double click on the **Simulate Behavioral Model** in the process window- this will launch the iSIM window as shown in Figure 29 The output 'y' has been highlighted in red for clarity.

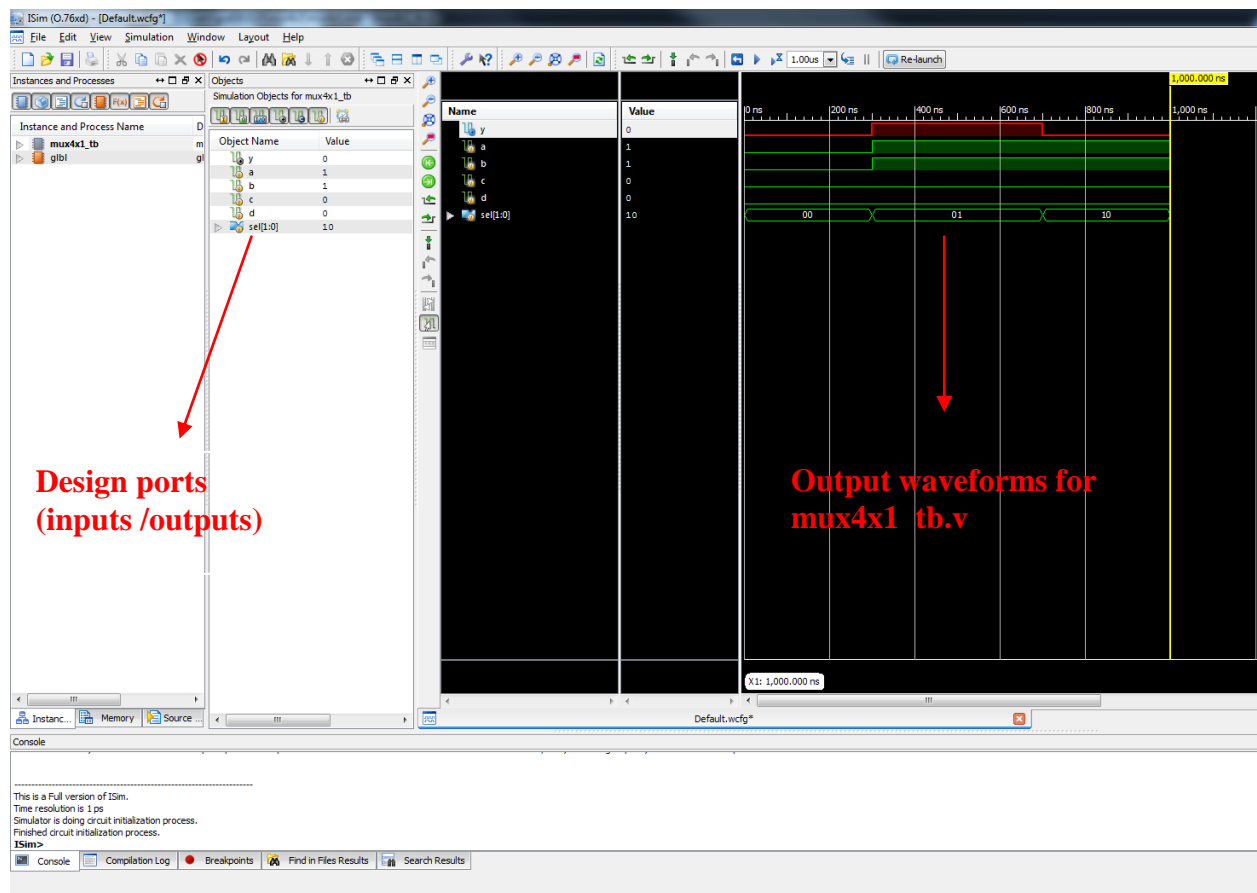


Figure 30: The iSIM window launched from Project Navigator

Behavioral simulation using ModelSim

You can also use ModelSim to simulate your design. To use ModelSim ensure that the Simulator field is set to Modelsim-PE Verilog in **Project→Design Properties** (you may change it back to iSIM if you so choose) as shown in Figure.

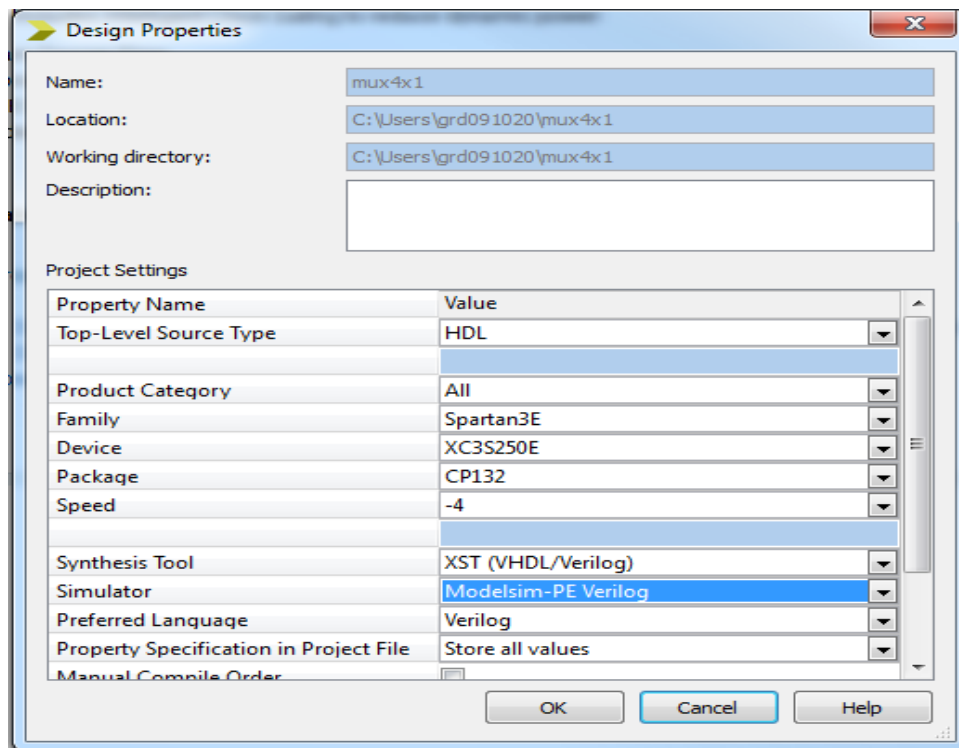


Figure 31: Changing the simulator field in Design Properties

Double clicking on the Simulate Behavioral Model will launch the ModelSim simulator interface.

In the Library window click on **Work** directory(all designs are saved in the Work directory) choose the test bench that you want to simulate (**mux4x1_tb**), right click and choose '**Simulate**'.

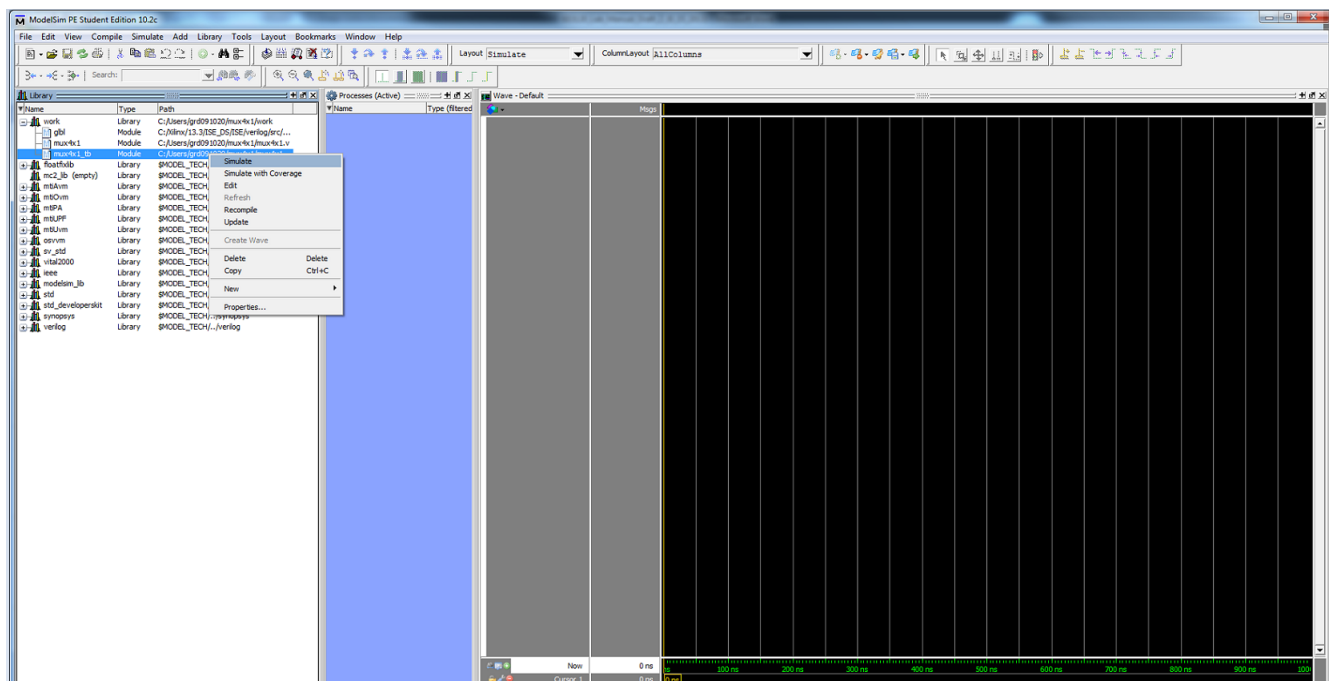


Figure 32: The ModelSim window

In the Sim window on the left, right click on the test bench name and choose ‘**Add wave**’

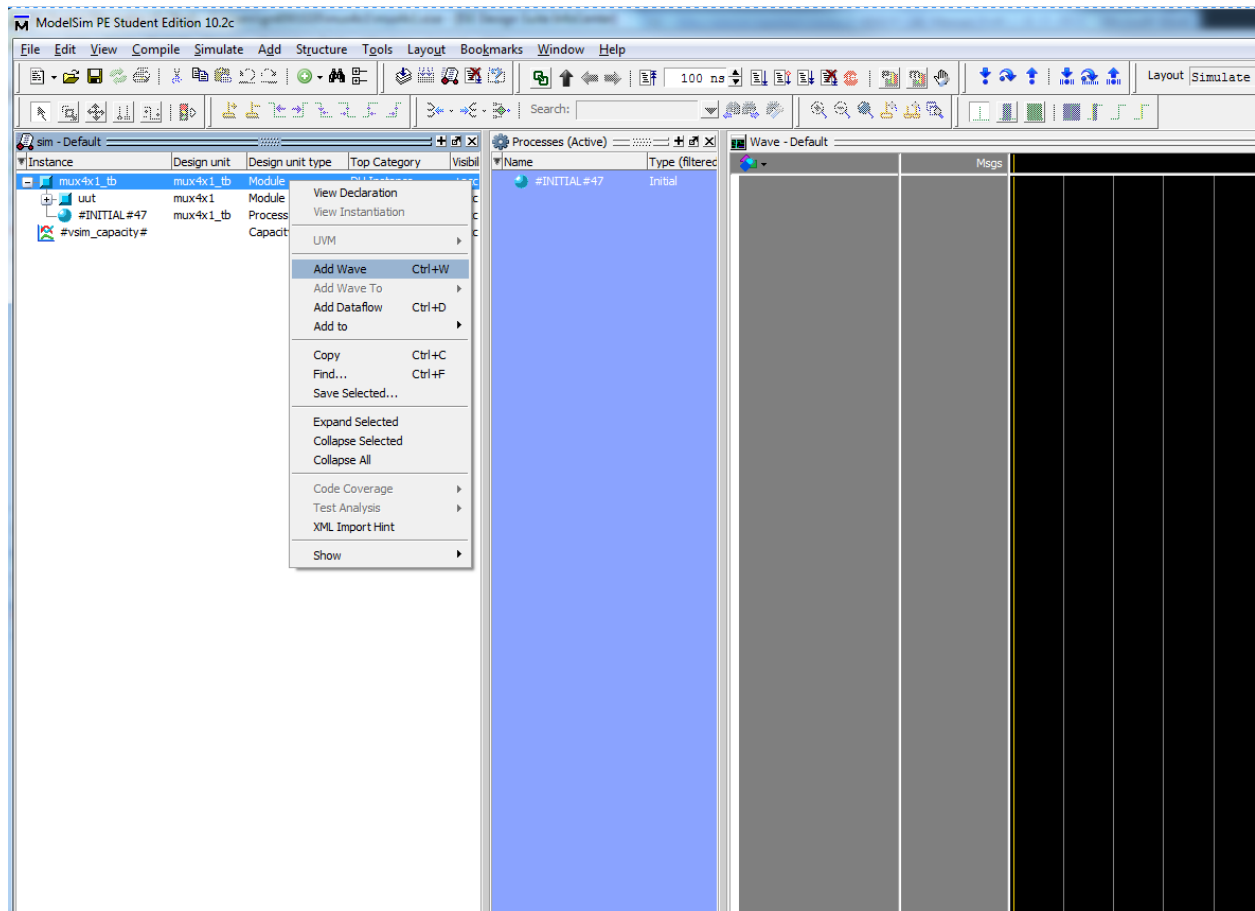


Figure 33: Adding test bench signals to the simulator

Click on **Run** or **Run All** to run the simulation.[Simulate →Run→Run -all]. You may choose to run the simulation for a specified time by typing the run command in the ModelSim prompt.

Appendix-A:

Verilog Hardware Modeling:

This is just an introductory level tutorial to the Verilog language. The reader is encouraged to go through the following Verilog tutorials to understand the language better:

- <http://www.asic-world.com/verilog/vbehave.html>
- <http://www.vol.webnexus.com/> [requires free registration]

1. Module:

A module is the basic building block in Verilog. It is defined as follows:

```
module <module_name> (<portlist>);  
    .  
    .          // module components  
    .  
endmodule
```

The <module_name> is the type of this module. The <portlist> is the list of connections, or ports, which allows data to flow into and out of modules of this type.

Verilog models are made up of modules. Modules, in turn, are made of different types of components. These include

- Parameters
- Nets
- Registers
- Primitives and Instances
- Continuous Assignments
- Procedural Blocks
- Task/Function definitions

2. Ports:

Ports are Verilog structures that pass data between two or more modules. Thus, ports can be thought of as wires connecting modules. The connections provided by ports can be either input, output, or bi-directional (inout).

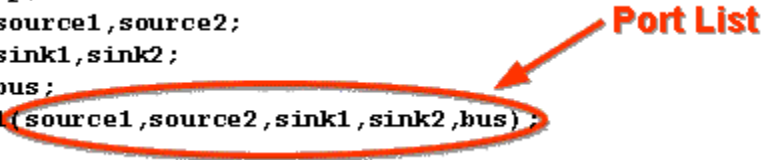
```
module foo(in1,in2,out1,out2,bidi):  
    input in1,in2;  
    output out1,out2;  
    inout bidi;  
    ...  
endmodule
```



Port List

Module instantiations also contain port lists. This is the means of connecting signals in the parent module with signals in the child module.

```
module top;
  wire source1,source2;
  wire sink1,sink2;
  wire bus;
  foo f1(source1,source2,sink1,sink2,bus);
  ...
endmodule
```



3. Nets:

Nets are the things that connect model components together. They are usually thought of as wires in a circuit. Nets are declared in statements like this:

```
net type [range] [delay3] list of net identifiers ;
```

Example:

```
wire w1, w2;
tri [31:0] bus32;
wire wire_number_5 = wire_number_2 & wire_number_3;
```

4. Registers:

Registers are storage elements. Values are stored in registers in procedural assignment statements. Registers can be used as the source for a primitive or module instance (i.e. registers can be connected to input ports), but they cannot be driven in the same way a net can.

Registers are declared in statements like this:

```
reg [range] list_of_register_identifiers ;
```

Example:

```
reg r1, r2;
reg [31:0] bus32;
```

5. Operators in Verilog:

Logical, arithmetic and relational operators available in Verilog are described in Table 1.

?: (conditional) [legal for real; associates right to left (others associate left to right)]
|| (logical or) [A smaller operand is zero-filled from its msb (0-fill); legal for real]
&& (logical and) [0-fill, legal for real]
| (bitwise or) ~| (bitwise nor) [0-fill]
^ (bitwise xor) ^~ ^~ (bitwise xnor, equivalence) [0-fill]
& (bitwise and) ~& (bitwise nand) [0-fill]
== (logical) != (logical) === (case) !== (case) [0-fill, logical versions are legal for real]
< (lt) <= (lt or equal) > (gt) >= (gt or equal) [0-fill, all are legal for real]
<< (shift left) >> (shift right) [zero fill; no -ve shifts; shift by x or z results in unknown]
+ (addition) - (subtraction) [if any bit is x or z for + - * / % then entire result is unknown]
* (multiply) / (divide) % (modulus) [integer divide truncates fraction; + - * / legal for real]

Verilog Unary Operators:

Operator	Name	Examples
!	logical negation	!123 is 'b0 [0, 1, or x for ambiguous; legal for real]
~	bitwise unary negation	~1'b10xz is 1'b01xx
&	unary reduction and	& 4'b1111 is 1'b1, & 2'bx1 is 1'bx, & 2'bz1 is 1'bx
~&	unary reduction nand	~& 4'b1111 is 1'b0, ~& 2'bx1 is 1'bx
	unary reduction or	Note:
~	unary reduction nor	Reduction is performed left (first bit) to right
^	unary reduction xor	Beware of the non-associative reduction operators
~^ ^~	unary reduction xnor	z is treated as x for all unary operators
+	unary plus	+2'bxz is +2'bxz [+m is the same as m; legal for real]
-	unary minus	-2'bxz is x [-m is unary minus m; legal for real]

Source: ASIC Design by Smith (<http://www-ee.eng.hawaii.edu/~msmith/ASICs/Files/pdf/CH11.3.pdf>)

Table 1 Verilog Operators

6. Continuous assignments:

Continuous assignments are sometimes known as data flow statements because they describe how data moves from one place, either a net or register, to another. They are usually thought of as representing combinational logic. In general, any logic functionality which can be implemented by means of a continuous assignment can also be implemented using primitive instances.

A continuous assignment looks like this:

```
assign [delay3] list_of_net_assignments ;
```

Examples:

```
assign w1 = w2 & w3;  
assign #1 mynet = enable; // mynet is assigned the value after 1 time unit.
```

7. Procedural Blocks:

Procedural blocks are the part of the language which represents sequential behavior. A module can have as many procedural blocks as necessary. These blocks are sequences of executable statements. The statements in each block are executed sequentially, but the blocks themselves are concurrent and asynchronous to other blocks.

There are two types of procedural blocks, initial blocks and always blocks.

```
initial <statement>          always <statement>
```

There may be many initial and always blocks in a module. Since there may be many modules in a model, there may be many initial and always blocks in the entire model. All initial and always blocks contain a single statement, which may be a compound statement, e.g.

```
initial  
begin statement1 ; statement2 ; ... end
```

a. Initial Block:

All initial blocks begin at time 0 and execute the initial statement. Because the statement may be a compound statement, this may entail executing lots of statements. There may be time or event controls, as well as all of the control constructs in the language. As a result, an initial block may cause activity to occur throughout the entire simulation of the model.

When the initial statement finishes execution, the initial block terminates. If the initial statement is a compound statement, then the statement finishes after its last statement finishes.

Example:

```
initial x = 0; // a simple initialization  
  
initial begin  
    x = 1;          // an initialization  
    y = f(x);  
#1 x = 0;          // a value change 1 time unit later  
    y = f(x);  
end
```


b. Always Block:

Always blocks also begin at time 0. The only difference between an always block and an initial block is that when the always statement finishes execution, it starts executing again. Note that if there is no time or event control in the always block, simulation time can never advance beyond time 0. Example,

```
always
    #10 clock = ~clock;
```

8. Behavioral modeling constructs:

a. Conditional if-else construct:

The if - else statement controls the execution of other statements in a procedural block.

Syntax:

```
if (condition)
    statements;

if (condition)
    statements;
else
    statements;

if (condition)
    statements;
else if (condition)
    statements;
.....
.....
else
    statements;
```

Example:

```
// Simple if statement
if (enable)
    q <= d;

// One else statement
if (reset == 1'b1)
    q <= 0;;
else
    q <= d;

// Nested if-else-if statements
if (reset == 1'b0)
    counter <= 4'b0000;
else if (enable == 1'b1 && up_en == 1'b1)
    counter <= counter + 1'b1;
else if (enable == 1'b1 && down_en == 1'b1);
    counter <= counter - 1'b0;
else
    counter <= counter; // Redundant code
```

b. Case statement:

The case statement compares an expression to a series of cases and executes the statement or statement group associated with the first matching case. Case statement supports single or multiple statements. Multiple statements can be grouped using begin and end keywords.

Syntax:

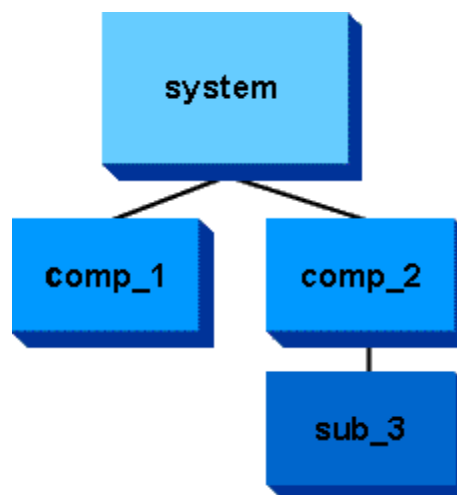
```
case (<expression>)
  <case1> : <statement>
  <case2> : <statement>
  .....
  default : <statement>
endcase
```

Example:

```
module mux (a,b,c,d,sel,y);
  input a, b, c, d;
  input [1:0] sel;
  output y;
  reg y;
  always @ (a or b or c or d or sel)
  case (sel)
    0 : y = a;
    1 : y = b;
    2 : y = c;
    3 : y = d;
    default : $display("Error in SEL");
  endcase
endmodule
```

9. Module instantiations and hierarchies:

Verilog allows you to represent the hierarchy of a design. A more common way of depicting hierarchical relationships is:



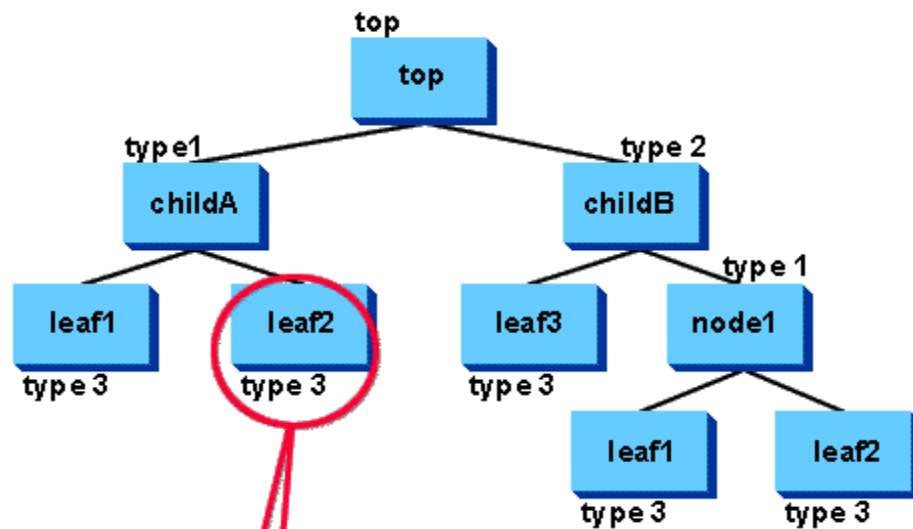
We say that a parent instantiates a child module. That is, it creates an instance of it to be a submodel of the parent. In this example,

system instantiates comp_1, comp_2

comp_2 instantiates sub_3

Modules in a hierarchy have both a type and a name. **Module types are defined in Verilog.** There can be many module instances of the same type of module in a single hierarchy. The module definition by itself does not create a module. Modules are created by being instantiated in another module, like this:

```
module <module_name_1> (<portlist>);  
    .  
    .  
    <module_name_2> <instance_name> (<portlist>);  
    .  
    .  
endmodule
```



```
module top;  
    type1 childA(ports...); //"port..." indicates a port list  
    type2 childB(ports...); //which will be explained later  
endmodule  
  
module type1(ports...);  
    type3 leaf1(ports...);  
    type3 leaf2(ports...);  
endmodule  
  
module type2(ports...);  
    type3 leaf3(ports...);  
    type1 node1(ports...);  
endmodule  
  
module type3(ports...);  
    //this module does not instantiate any other modules  
endmodule
```

Appendix-B:

This section contains some useful information on the Digilent BASYS board that will be used in this lab. For further details please refer to the [Digilent user's manual](#).

B.1 Spartan 3E pin definitions

Basys2 Spartan-3E pin definitions											
Pin	Signal	Pin	Signal	Pin	Signal	Pin	Signal	Pin	Signal	Pin	Signal
C12	JD1	P11	SW0	N14	CC	B2	JA1	P8	MODE0	M7	GND
A13	JD2	M2	USB-DB1	N13	DP	C2	USB-WRITE	N7	MODE1	P5	GND
A12	NC	N2	USB-DB0	M13	AN2	C3	PS2D	N6	MODE2	P10	GND
B12	NC	M9	NC	M12	CG	D1	NC	N12	CCLK	P14	GND
B11	NC	N9	NC	L14	CA	D2	USB-WAIT	P13	DONE	A6	VDDO-3
C11	BTN1	M10	NC	L13	CF	L2	USB-DB4	A1	PROG	B10	VDDO-3
C6	JB1	N10	NC	F13	RED2	L1	USB-DB3	N8	DIN	E13	VDDO-3
B6	JB2	M11	LD1	F14	GRN0	M1	USB-DB2	N1	INIT	M14	VDDO-3
C5	JB3	N11	CD	D12	JD4	L3	SW1	P1	NC	P3	VDDO-3
B5	JA4	P12	CE	D13	RED1	E2	SW6	B3	GND	M8	VDDO-3
C4	NC	N3	SW7	C13	JD3	F3	SW5	A4	GND	E1	VDDO-3
B4	SW3	M6	UCLK	C14	RED0	F2	USB-ASTB	A8	GND	J2	VDDO-3
A3	JA2	P6	LD3	G12	BTN0	F1	USB-DSTB	C1	GND	A5	VDDO-2
A10	JC3	P7	LD2	K14	AN2	G1	LD7	C7	GND	E12	VDDO-2
C9	JC4	M4	BTN2	J12	AN1	G3	SW4	C10	GND	K1	VDDO-2
B9	JC2	N4	LD5	J13	BLU2	H1	USB-DB6	E3	GND	P9	VDDO-2
A9	JC1	M5	LD0	J14	HSYNC	H2	USB-DB5	E14	GND	A11	VDDO-1
B8	MCLK	N5	LD4	H13	BLU1	H3	USB-DB7	G2	GND	D3	VDDO-1
C8	RCCLK	G14	GRN2	H12	CB	B14	TMS	H14	GND	D14	VDDO-1
A7	BTN3	G13	GRN1	J3	JA3	B13	TCK-FPGA	J1	GND	K2	VDDO-1
B7	JB4	F12	AN0	K3	SW2	A2	TDO-USB	K12	GND	L12	VDDO-1
P4	LD6	K13	VSYNC	B1	PS2C	A14	TDO-S3	M3	GND	P2	VDDO-1

FPGA pin definition table color key		
Grey		Not available to user
Green		User I/O devices
Yellow		Data ports
Tan		Pmod connector signals
Blue		USB signals

B.2 Testing the Digilent board

The Digilent Adept tool can be used to test the functionality of a BASYS board as follows:

- Connect the board to the USB and set the power switch to ON.
- Run Digilent Adept
- Click on Test → Start Test

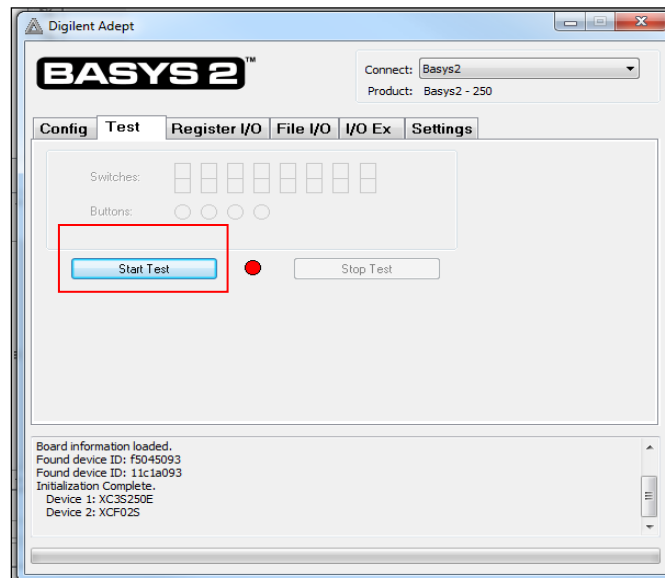


Figure B.1 The Digilent Adept test function

- The tool shows you the current switch configuration and the display counts 0000→1111→FFFF

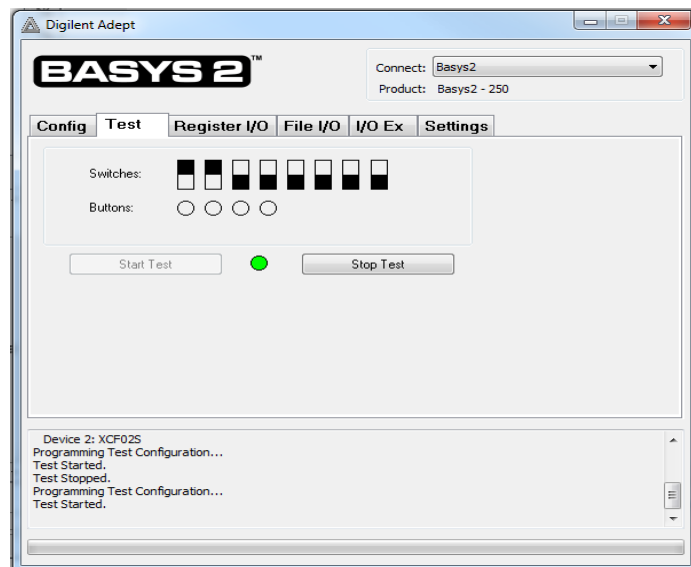


Figure B.2 Testing the Digilent board